

AD-A244 400



②



Large Deformation Analysis of Nonlinear Homogeneous and Heterogeneous Media Using an Arbitrary Lagrangian-Eulerian Finite Element Method

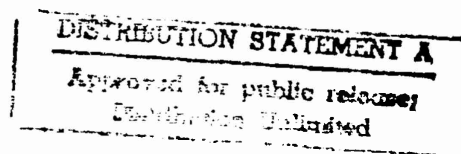
Somnath Ghosh
Department of Engineering Mechanics

DTIC
ELECTE
JAN 10 1992
S B D

U.S. Army Research Office
Research Triangle Park, North Carolina 27709-2211

Grant No. DAAL03-91-G-0069
Final Report

November 1991



92-00686



92 1 8 076



Large Deformation Analysis of Nonlinear Homogeneous and Heterogeneous Media Using an Arbitrary Lagrangian-Eulerian Finite Element Method

Somnath Ghosh
Department of Engineering Mechanics

U.S. Army Research Office
Research Triangle Park, North Carolina 27709-2211

Grant No. DAAL03-91-G-0069
Final Report
RF Project No. 769182/724944

November 1991

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
<small>Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.</small>				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE November 12, 1991		3. REPORT TYPE AND DATES COVERED Final March 15, 1991 - August 15, 1991
4. TITLE AND SUBTITLE Large Deformation Analysis of Nonlinear Homogeneous and Heterogeneous Media Using an Arbitrary Lagrangian-Eulerian Finite Element Method			5. FUNDING NUMBERS DAAL 03-91-G-0069	
6. AUTHOR(S) Dr. Somnath Ghosh				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) The University of Alabama Tuscaloosa, AL 35487			8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) U. S. Army Research Office P. O. Box 12211 Research Triangle Park, NC 27709-2211			10. SPONSORING/MONITORING AGENCY REPORT NUMBER ARO 29317.1-E6	
11. SUPPLEMENTARY NOTES The view, opinions and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy, or decision, unless so designated by other documentation.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution unlimited.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) In this paper, a new finite element formulation has been developed for media in which the second phase particulates or unidirectional fibers are randomly dispersed in a matrix. Deficiencies of the conventional mesh generators can be overcome by discretizing the domain using the method of Dirichlet tessellation. This method generates convex polygons known as Voronoi polygons which are treated as elements in a finite element analysis. An assumed stress hybrid method has been used to formulate the element stiffness matrix. Effect of the second phase has been incorporated in each element using a self-consistent type method. Several numerical examples have been conducted to validate the effectiveness of the model.				
14. SUBJECT TERMS			15. NUMBER OF PAGES 40	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED	18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT UNCLASSIFIED	20. LIMITATION OF ABSTRACT UL	

REPORT OF INVENTIONS AND SUBCONTRACTS

(Pursuant to "Patent Rights" Contract Clause) (See Instructions on Reverse Side.)

Form Approved
OMB No. 0704-0097
Expires Jun 30, 1983

Public reporting burden for this collection of information is estimated to average 5 minutes per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0097), Washington, DC 20503.

1a. NAME OF CONTRACTOR/SUBCONTRACTOR		1b. NAME OF GOVERNMENT PRIME CONTRACTOR		3. TYPE OF REPORT (If used)	
University of Alabama		Army Research Office		a. INTERIM <input type="checkbox"/> b. FINAL <input checked="" type="checkbox"/>	
2a. ADDRESS (Include ZIP Code)		2b. ADDRESS (Include ZIP Code)		4. REPORTING PERIOD (YYMMDD)	
Box 870104				a. FROM 91-03-15	
Tuscaloosa, AL 35487-0104				b. TO 91-08-15	

SECTION I - SUBJECT INVENTIONS

5. "SUBJECT INVENTIONS" REQUIRED TO BE REPORTED BY CONTRACTOR/SUBCONTRACTOR (If "None," so state)

a. NAME(S) OF INVENTION(S) (Last, First, MI)	b. TITLE OF INVENTION(S)	c. DISCLOSURE NO., PATENT APPLICATION SERIAL NO. OR PATENT NO.	4. ELECTION TO FILE PATENT APPLICATIONS				e. CONFIRMATORY INSTRUMENT OR ASSIGNMENT FORWARDED TO CONTRACTING OFFICER
			(1) United States	(2) Foreign	(a) Yes	(b) No	
Somnath Ghosh	A New Finite Element Method For Random Composites						(1) Yes <input type="checkbox"/> (2) No <input type="checkbox"/>

6. EMPLOYER OF INVENTION(S) NOT EMPLOYED BY CONTRACTOR/SUBCONTRACTOR

(1) (a) Name of Inventor (Last, First, MI)	(2) (a) Name of Inventor (Last, First, MI)	(1) Title of Invention	(2) Foreign Countries of Patent Application
(b) Name of Employer	(b) Name of Employer		
(c) Address of Employer (Include ZIP Code)	(c) Address of Employer (Include ZIP Code)		

SECTION II - SUBCONTRACTS (Containing a "Patent Rights" clause)

6. SUBCONTRACTS AWARDED BY CONTRACTOR/SUBCONTRACTOR (If "None," so state)

a. NAME OF SUBCONTRACTOR(S)	b. ADDRESS (Include ZIP Code)	c. SUBCONTRACT NO.(S)	d. DFRAT "PATENT RIGHTS"		e. DESCRIPTION OF WORK TO BE PERFORMED UNDER SUBCONTRACT(S)	f. SUBCONTRACT DATES (YYMM)	
			(1) Clause Number	(2) Date (YYMMDD)		(1) Award	(2) Expiry Complete

SECTION III - CERTIFICATION

7. CERTIFICATION OF REPORT BY CONTRACTOR/SUBCONTRACTOR		8. CERTIFICATION OF REPORT BY CONTRACTOR/SUBCONTRACTOR	
a. NAME OF AUTHORIZED CONTRACTOR/SUBCONTRACTOR OFFICIAL (Last, First, MI)		a. NAME OF AUTHORIZED CONTRACTOR/SUBCONTRACTOR OFFICIAL (Last, First, MI)	
b. TITLE		b. TITLE	
c. I certify that the reporting party has procedures for prompt identification and timely disclosure of "Subject Inventions," that such procedures have been followed and that all "Subject Inventions" have been reported.		c. I certify that the reporting party has procedures for prompt identification and timely disclosure of "Subject Inventions," that such procedures have been followed and that all "Subject Inventions" have been reported.	
d. SIGNATURE		d. SIGNATURE	
e. DATE SIGNED		e. DATE SIGNED	

11/14/91

1. FOREWORD

The project entitled " Large Deformation Analysis of Nonlinear Homogeneous and Heterogeneous Media Using an Arbitrary Lagrangian-Eulerian Finite Element Method " (Grant No. DAAL03-91-G-0069) has been transferred from the University of Alabama to the Ohio State University effective August 16, 1991. The principal investigator of the project, Dr. Somnath Ghosh has assumed a faculty position at the Ohio State University and the project is continuing at OSU. Therefore, in this report only the first part of the ongoing work is presented. Subsequent progresses will be reported from the Ohio State University.

2. LIST OF APPENDICES

First draft of a manuscript entitled " ANALYSIS OF RANDOM COMPOSITES USING VORONOI CELL FINITE ELEMENTS"

3. BODY OF THE PROBLEM STUDIED

A. STATEMENT OF THE PROBLEM

In this part of the study, a new finite element technique has been developed for macroscopic analysis of composites with a stochastic dispersion of microscopic heterogeneities. In these composites, particles or fibers of various shapes or sizes may be randomly scattered within the matrix. The present work introduces a *material based discretization* system that adequately accounts for the morphology of the heterogeneous domain and then formulates a finite element scheme for accommodating multi-sided elements with second phase materials present.

B. SUMMARY OF MOST IMPORTANT RESULTS

The heterogeneous domain with random distribution of second phase has been discretized by Dirichlet tessellation so that each element contains only one inclusion. The resulting network of Voronoi cells are now assumed to be elements in a finite element formulation.

Indeed, the multi-noded Voronoi cells are rather unconventional elements and a displacement based formulation with such elements is extremely difficult due to the varying number of sides in each element. An assumed stress hybrid finite element formulation has proven to be extremely successful in this process. Several different test cases have been experimented with for linear elastic problems and good correlation between numerical solutions and analytical results have been obtained.

The effect of the second phase has been incorporated by the introduction of a transformation strain in the element level in a fashion similar to self-consistent schemes. The results have been compared with conventional finite element predictions and are in reasonable agreement. Work is now in progress for refining this model.

C. PUBLICATIONS

Two manuscripts are presently being written and will be forwarded to ARO as soon as they are completed.

D. DEGREES AWARDED FROM THIS PROJECT

No degrees have been awarded so far from this project

4. REPORT OF INVENTIONS

A New Finite element formulation for random composites using Voronoi Cell elements



<input checked="checked" type="checkbox"/>	
<input type="checkbox"/>	
<input type="checkbox"/>	
203	
70P	
Dist	
A-1	

APPENDIX

ANALYSIS OF RANDOM COMPOSITES USING VORONOI CELL FINITE ELEMENTS

SOMNATH GHOSH

**Department of Engineering Mechanics
The Ohio State University**

(preliminary draft of a manuscript being prepared for submission)

Table of Contents

Item	Page Number
1. Introduction	1
2. Formulation for voronoi cell finite elements by Mallett's method	2
The least square finite element scheme	2
The finite difference scheme	3
Rank deficiency of stiffness matrix	4
Numerical examples	4
3. Proposed formulation by assumed stress hybrid method	8
Element formulation	8
Invariance stiffness matrix	12
Numerical examples	13
4. Modelling of second phase material present in voronoi cell	26
Problem formulation	26
Numerical examples	30
5. References	36

1 Introduction

Research related to the study of composite behavior during the last two decades, has resulted in a number of computational models for their analysis. Most of these models use the *homogenization method* for reflecting the influence of material microstructure on the macroscopic behavior. These methods are based on asymptotic analyses, which predominantly make assumptions of periodic repetition of microstructures. However, in practice, there exists a wide variety of composite samples with a stochastic dispersion of microscopic heterogeneities. In these composites, particles or fibers of various shapes and sizes may be randomly scattered within the matrix and even be clustered in certain regions. The distribution of shapes, sizes and the spatial coordinates of the second phase has a profound influence on the mechanical behavior of the overall structure and therefore, must be taken into account in a rigorous analysis.

A fundamental requirement in the development of finite element models for analysis of the above class of random composites is the generation of a robust mesh, that will adequately account for the morphology of the composite domain under consideration. The Dirichlet tessellation seems to provide an excellent foundation for natural evolution of discretization process while accounting for the microstructure. This is a method of subdividing an Euclidean space into n -dimensional bounded convex polytopes. It may be perceived as production of a network of interfaces, formed by the impingement of expanding hyperspheres about random nuclei that are growing at a uniform rate from zero. If the second phase particles are realized as points in space, the convex polytopes (polygons in two dimensions) known as Voronoi polytopes resulting from this discretization, would encompass one inclusion each at most. These Voronoi polygons, in two dimension, are used as elements in a finite element analysis of random composites.

2 Formulation of Voronoi Cell Finite Elements by Mallett's Method

Mallett [1] proposed a least square finite element scheme and a finite difference element scheme for the formulation of voronoi polygon elements. In the first scheme, the displacement components in each element is represented as linear function of position and then they are determined by giving a least square fit to the nodal values of the displacement components. The latter scheme completely eliminates the need to assume any sort of approximation for the displacement field in the interior of the element. Instead, the element's average displacement gradient components are calculated to form the strain displacement matrix.

2.1 The Least Square Finite Element Scheme:

Displacement field in a n-sided polygonal element that represents the rigid body displacements and the constant strain field, can be expressed as follows

$$\begin{aligned}u(x, y) &= \alpha_1 + \alpha_2 x + \alpha_3 y \\v(x, y) &= \beta_1 + \beta_2 x + \beta_3 y\end{aligned}\tag{1}$$

where x and y are global Cartesian coordinates.

Evaluating the element displacement components at the i -th node and requiring that these values coincide with the nodal displacement components u_i and v_i leads to the following equations

$$\begin{Bmatrix} u_1 \\ \cdot \\ \cdot \\ \cdot \\ u_n \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot \\ 1 & x_n & y_n \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}$$

or

$$\{\mathbf{u}\} = [\mathbf{A}]\{\boldsymbol{\alpha}\}\tag{2}$$

As the number of nodes of an element will be, in most, cases more than three, equations (2) constitutes an overdetermined set of equations. Applying least square method to solve the overdetermined system, one obtains

$$\{\boldsymbol{\alpha}\} = ([\mathbf{A}]^T[\mathbf{A}])^{-1}[\mathbf{A}]^T\{\mathbf{u}\}\tag{3}$$

Substituting this expression for $\{\alpha\}$ and the similar expression for $\{\beta\}$ into eq. (1) gives

$$\begin{aligned} u(x, y) &= [1 \quad x \quad y]([A]^T[A])^{-1}[A]^T\{u\} = N\{u\} \\ v(x, y) &= [1 \quad x \quad y]([A]^T[A])^{-1}[A]^T\{v\} = N\{v\} \end{aligned} \quad (4)$$

The components of strain can be related to nodal displacement components by

$$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N}{\partial x} & 0 \\ 0 & \frac{\partial N}{\partial y} \\ \frac{\partial N}{\partial y} & \frac{\partial N}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

or

$$\{\epsilon\} = [B]\{d\} \quad (5)$$

where $[B]$ is the strain displacement matrix and $\{d\}$ is the element nodal displacement vector. It can be easily seen that $\frac{\partial N}{\partial x}$ and $\frac{\partial N}{\partial y}$ is constant over the element and so is matrix $[B]$. The element stiffness matrix, for unit thickness, thus can be expressed as

$$[K]_e = \int_v [B]^T [E] [B] dv = A_e [B]^T [E] [B] \quad (6)$$

where A_e is the area of the element and $[E]$ is the elasticity matrix.

The nodal forces implied by the above stiffness matrix are not statically equivalent to the edge tractions implied by the stresses within the element. To rectify this deficiency a $[S]$ matrix of size $2n \times 3$ is determined that transforms the stress components into statically equivalent nodal force components.

$$\{F\}_e = [S]\{\sigma\} = [S][E][B]\{d\} \quad (7)$$

from which element stiffness matrix can be written as

$$[K]_e = [S][E][B] \quad (8)$$

which in general is not symmetric.

2.2 The Finite Difference Element Scheme :

In this scheme, the average of displacement gradients $\frac{\partial u}{\partial x}, \frac{\partial u}{\partial y}$ and $\frac{\partial v}{\partial x}, \frac{\partial v}{\partial y}$ over an element is calculated and written as

$$\begin{aligned} \frac{\partial \bar{u}}{\partial x} &= \frac{a^T u}{A_e} & \frac{\partial \bar{u}}{\partial y} &= \frac{b^T u}{A_e} \\ \frac{\partial \bar{v}}{\partial x} &= \frac{a^T v}{A_e} & \frac{\partial \bar{v}}{\partial y} &= \frac{b^T v}{A_e} \end{aligned}$$

where

$$\begin{aligned} a_i &= \frac{1}{2}(y_{i+1} - y_{i-1}) \\ b_i &= \frac{1}{2}(x_{i-1} - x_{i+1}) \quad i = 1 \dots n \end{aligned}$$

with the understanding that (x_0, y_0) and (x_{n+1}, y_{n+1}) are to be interpreted as (x_n, y_n) and (x_1, y_1) respectively.

Assuming a state of constant strain within the element, element's strain component averages are then given by

$$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{1}{A_e} \begin{bmatrix} a^T & 0^T \\ 0^T & b^T \\ b^T & a^T \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

or

$$\{\epsilon\} = [B]\{d\} \quad (9)$$

The matrix $[B]$ is constant over the element, hence the stiffness matrix can be expressed as

$$[K]_e = A_e [B]^T [E] [B] \quad (10)$$

2.3 Rank Deficiency of Stiffness Matrix

The element stiffness matrix produced by either of the schemes has a rank of three, irrespective of the number of nodes n in an element, and its rank deficiency is $2n-6$. Assemblage of rank deficient stiffness matrices produces a rank deficient global stiffness matrix. Mailett proposed two different methods for control of hourglass or zero energy nodes produced by a rank deficient stiffness matrix. In the first method an artificial stiffness is added at the element stiffness matrix level, which will inhibit hourglass deformation without seriously affecting non-hourglass deformation. In the second method a constraint is imposed at the global stiffness level, by means of the Lagrange multiplier technique, to eliminate rather than suppress hourglass deformation. However elimination of hourglass modes at the element level leads to a global stiffness matrix that is excessively stiff. Elimination of hourglass deformation at the global level has been found to be the only viable alternative and the following numerical examples are based on the same principle.

2.4 Numerical Examples

The material properties assumed here are used in all subsequent examples unless otherwise specified. Further the problems are always solved under plain strain condition. Poisson's ratio is assumed as 0.3 and Young's modulus as 2 KN/sq.mm. The value of Young's modulus

is unimportant as it merely serves as a scale factor for the displacement.

Stretching Problem:

A square material body of size 40 mm x 40 mm is discretized into 24 voronoi polygons (Fig. 1) and is subjected to a uniform tensile force of 100 KN. Analysis of this problem by Mallett's method yields end deflection of 4.55 mm in X direction and σ_x of 2.5 kN/sq.mm in individual elements which are the exact theoretical values. Also the displacement of the nodes in X direction versus their position from the fixed end has been found to be linearly proportional (Fig. 2).

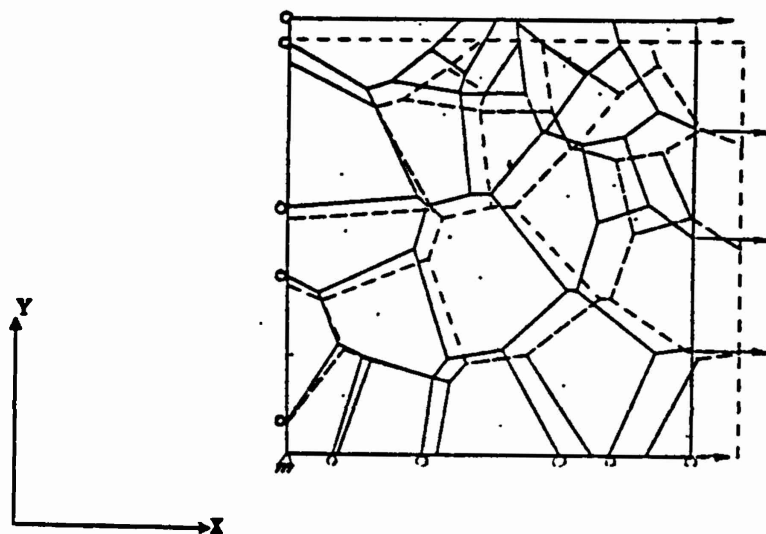


Fig. 1. Body subjected to tensile load.

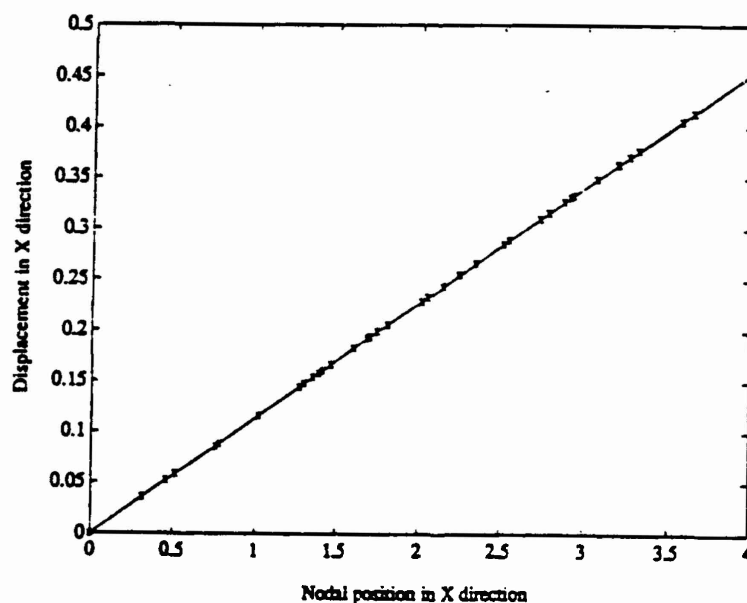


Fig. 2. Nodal displacement in X direction vs. nodal position.

Bending Problem:

A material body of size 40 mm x 160 mm is used as a cantilever beam with one end fixed and other end subjected to pure moment of 40 KN-mm. The beam is discretized into 28 voronoi polygon elements (Fig. 3). Longitudinal stress σ_x along a transverse section is calculated by the above method and is plotted as shown in Fig. 4. The stresses are found to be constant within an element and differ widely from theoretical value.

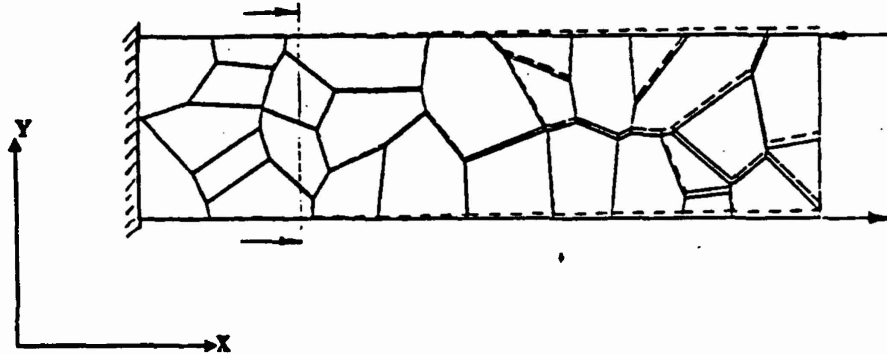


Fig. 3. Beam subjected to pure end moment.

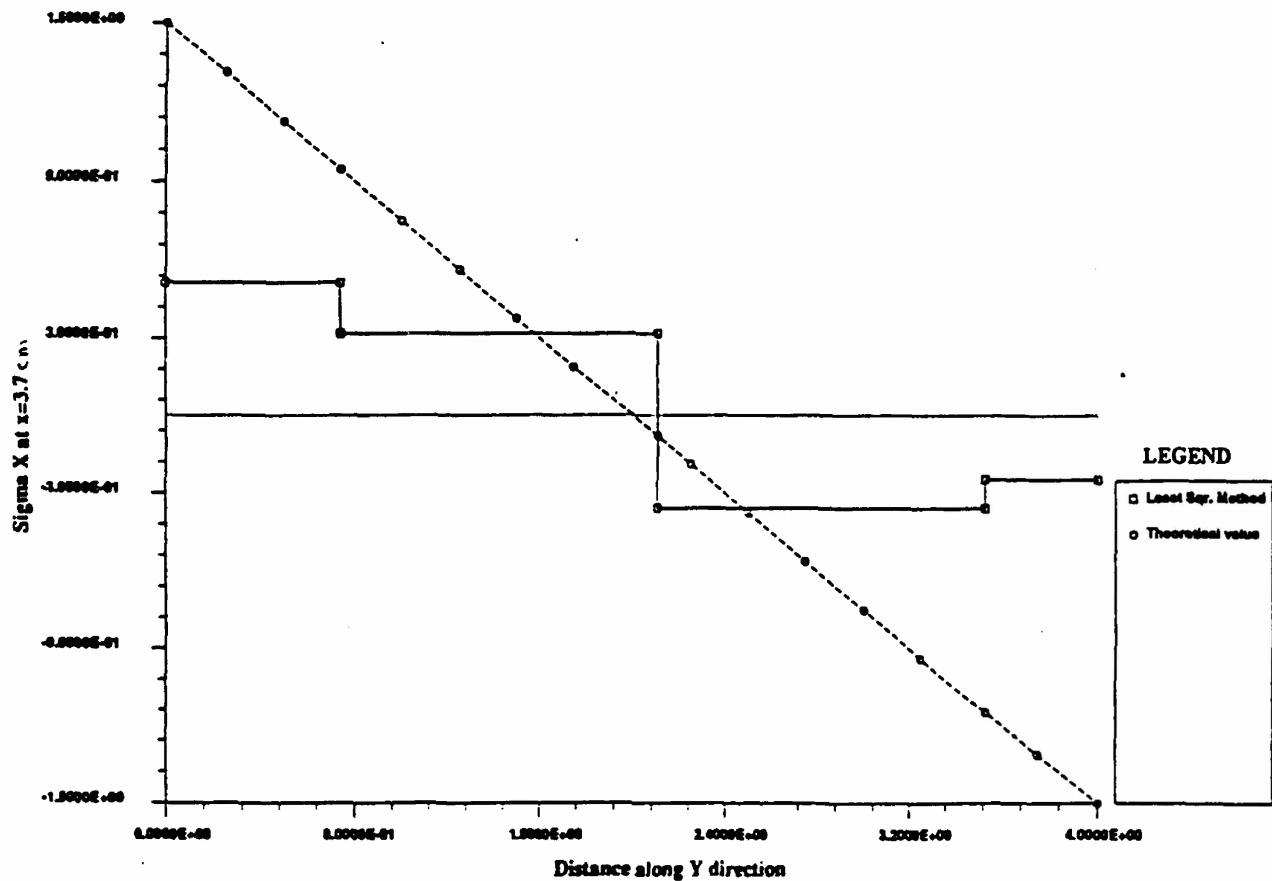


Fig. 4. Sigma X along a transverse section.

The calculated tip deflection is 1.546 mm, which is only 35% of the theoretical value of 4.368 mm. Fig. 5 shows a plot of the vertical displacement of the nodes at the top surface against the theoretical values.

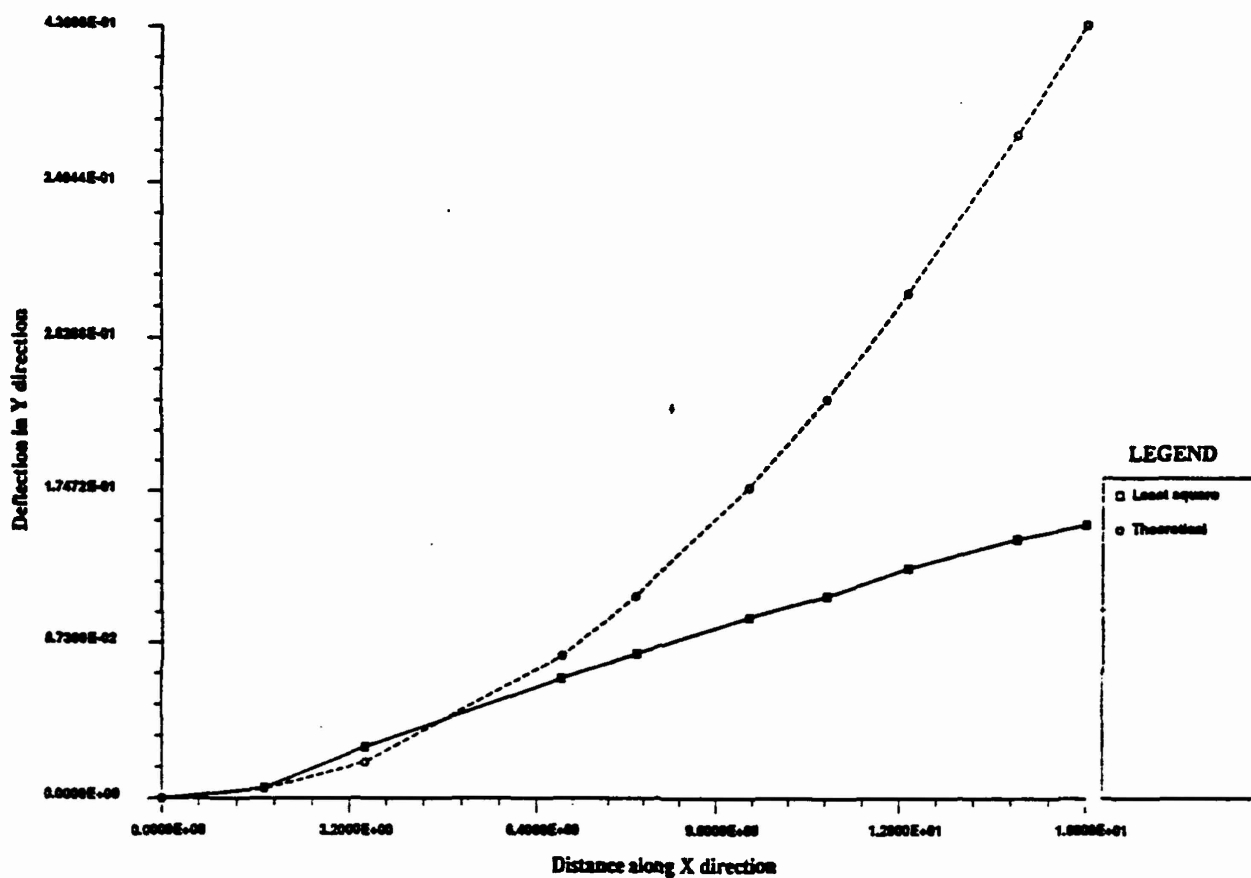


Fig. 5. Deflection of top surface.

3 Proposed Formulation by Assumed Stress Hybrid Method

Large amount of error in predicting the deflection and stress of a cantilever beam under pure end moment by Mallett's method was the motivation to look for an alternative formulation. In the earlier section it was shown that a least square fit of the displacement field from the nodal values produce a rank deficient stiffness matrix for elements having more than three nodes. Even if the rank deficiency is taken care of by global hourglass control, the elements exhibit constant stress which are widely different from theoretical values. Pian [2] suggested an alternative derivation of the element stiffness matrices by assumed stress hybrid method. In the proposed method, instead of a required continuous displacement function over the element, it is necessary only to write down the boundary displacements that will guarantee a complete displacement compatibility. The assumed-stress hybrid method is a way of formulating a stiffness matrix by use of independent assumptions of (a) an equilibrium stress field within the element, and (b) interelement-compatible displacement nodes on the element boundary. As this method necessitates the interpolation of displacement field not in the interior of the element but only along the element boundary, it becomes easy to handle polygonal elements having any number of nodes.

3.1 Element Formulation

The derivation is based on the principle of minimum complementary energy, for which the functional to be varied is

$$\pi_c = \int_v \frac{1}{2} S_{ijkl} \sigma_{ij} \sigma_{kl} dv - \int_{\partial v} T_i \bar{u}_i ds \quad (11)$$

where S_{ijkl} is the elastic compliance, \bar{u}_i are the prescribed displacements along the boundary ∂v and T_i are the boundary tractions that are related to σ_{ij} .

The stresses satisfy the equilibrium condition

$$\frac{\partial \sigma_{ij}}{\partial x_j} + \bar{F}_i = 0 \quad (12)$$

and is compatible with the prescribed boundary tractions. In applying the finite element method, the assumed stress field need not be continuous across the interelement boundaries, but equilibrium must be maintained for the surface tractions T_i , defined by

$$T_i = \sigma_{ij} n_j \quad (13)$$

where n_j are the components of the unit vector normal to the boundary.

In the application of this variational principle, one begins with a stress field that satisfies the differential equations of equilibrium. The stress distribution can be expressed as

$$\{\sigma\} = [P]\{\beta\} \quad (14)$$

where $\{\beta\}$ is the column of m undetermined stress coefficients $\beta_1, \beta_2 \dots \beta_m$ and terms of the matrix $[P]$ are functions of coordinates x_i, y_i .

The surface tractions T_i have been expressed in terms of the stress σ_{ij} by equation (13) and hence can be related to the undetermined stress coefficients $\{\beta\}$ as

$$\{T\} = [R]\{\beta\} \quad (15)$$

Also, the prescribed boundary displacements \bar{u}_i can be interpolated from generalized displacement $\{q\}$ at the nodes, in the form

$$\{u\} = [L]\{q\} \quad (16)$$

where the terms in the matrix $[L]$ contain coordinates on the surface.

The equation (11) can be rewritten as

$$\pi_c = \int_v \frac{1}{2} \{\sigma\}^T [S] \{\sigma\} dv - \int_{\partial v} [T]^T \{u\} ds \quad (17)$$

Substituting the expressions for σ, T and u one obtains

$$\pi_c = \frac{1}{2} \{\beta\}^T [H] \{\beta\} - \{\beta\}^T [G] \{q\} \quad (18)$$

$$\text{where } [H] = \int_v [P]^T [S] [P] dv \quad (19)$$

$$\text{and } [G] = \int_{\partial v} [R]^T [L] ds \quad (20)$$

Making π_c stationary with respect to stress parameters

$$\frac{\partial \pi_c}{\partial \beta_i} = 0 \quad \text{for } i = 1, 2, \dots, m \quad (21)$$

gives

$$[H]\{\beta\} = [G]\{q\} \quad \text{or} \quad \{\beta\} = [H]^{-1}[G]\{q\} \quad (22)$$

Substituting $\{\beta\}$ in the expression for complementary strain energy for an element of volume v , one obtains

$$\begin{aligned} \mathcal{U} &= \int_v \frac{1}{2} \{\sigma\}^T [S] \{\sigma\} dv \\ &= \frac{1}{2} \{\beta\}^T [H] \{\beta\} \end{aligned}$$

$$= \frac{1}{2} \{q\}^T [G]^T [H]^{-1} [G] \{q\}$$

$$= \frac{1}{2} \{q\}^T [K] \{q\}$$

where $[K] = [G]^T [H]^{-1} [G]$ is the stiffness matrix.

The stiffness matrix $[K]$ will be rank deficient if it's rank is less than $n-l$ where n is the number of degrees of freedom and l is the number of rigid body nodes. A necessary condition for the resulting stiffness matrix to have sufficient rank is $m \geq n-l$, where m is the number of independent β -stress parameters. Because the derivation of the element stiffness matrix involves inversion of $[H]$ matrix which is of the order $m \times m$, it is advantageous to use as minimum a number of β -parameters as possible. There are also indications that overuse of β -parameters will yield over-rigid elements. An ideal situation is to choose the stress terms such that m is equal to $n-l$. However, it is not always possible to adhere to this as the number of β -terms is also governed by the fact that the assumed stress polynomial should have as many terms as required to satisfy the equilibrium equation (12). Hence for some elements, number of β -terms may be more than $n-l$. The resulting element properties by this approach will in general not be invariant. However, if an optimal local reference coordinate system is used in the element formulation, invariance may be achieved.

To illustrate the assumed stress hybrid method, components of the element stiffness matrix are derived as follows.

The stress functions, involving 5 β -terms, that satisfy the equation of equilibrium may be assumed as follows

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & y & 0 & 0 & 0 \\ 0 & 0 & 1 & x & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \cdot \\ \cdot \\ \cdot \\ \beta_5 \end{Bmatrix} = [P] \{ \beta \}$$

This is suitable for elements having up to 4 nodes. However for elements with higher number of nodes, as in the case of voronoi elements, it is necessary to increase the number of β -terms to get rank sufficiency of the stiffness matrix. The stress polynomial $[P]$ for 10 β terms is given as

$$\begin{bmatrix} 1 & y & 0 & 0 & 0 & x & 0 & y^2 & 0 & x^2 \\ 0 & 0 & 1 & x & 0 & 0 & y & 0 & x^2 & y^2 \\ 0 & 0 & 0 & 0 & 1 & -y & -x & 0 & 0 & -2xy \end{bmatrix}$$

and for 17 β terms as

$$\begin{bmatrix} 1 & y & 0 & 0 & 0 & x & 0 & y^2 & 0 & x^2 & xy & 0 & x^3 & y^3 & 3x^2y & xy^2 & 0 \\ 0 & 0 & 1 & x & 0 & 0 & y & 0 & x^2 & y^2 & 0 & xy & 3xy^2 & 0 & y^3 & x^3 & x^2y \\ 0 & 0 & 0 & 0 & 1 & -y & -x & 0 & 0 & -2xy & -y^2/2 & -x^2/2 & -3x^2y & 0 & -3xy^2 & -y^3/3 & -x^3/3 \end{bmatrix}$$

The stress polynomial involving 17 β -terms is thus suitable for elements having up to 10 nodes (number of degrees of freedom 2×10 , minus number of rigid body nodes 3, dictates a minimum of 17 β -terms). Hence for elements having up to 4 nodes, 5 β -terms are used; for elements having more than 4 but less than or equal to 6 nodes, 10 β -terms are used; for elements having more than 6 but less than or equal to 10 nodes, 17 β terms are used.

For the i th side of an element

$$\{u\} = [L]\{q\} = \begin{bmatrix} 1 - a/l_i & 0 & a/l_i & 0 \\ 0 & 1 - a/l_i & 0 & a/l_i \end{bmatrix} \begin{Bmatrix} q_{2i-1} \\ q_{2i} \\ q_{2i+1} \\ q_{2i+2} \end{Bmatrix}$$

where l_i is the length of side i and a is the distance measured from node i to node $i+1$. The assumed boundary displacement variation is linear. Therefore, the interelement compatibility is assured if the nodal displacements of adjacent elements are matched.

Traction on the side i is given as (for the sake of simplicity only 5 β -terms are considered)

$$\{T\}_i = \begin{bmatrix} \sigma_x n_{i1} + \tau_{xy} n_{i2} \\ \tau_{xy} n_{i1} + \sigma_y n_{i2} \end{bmatrix} = \begin{bmatrix} n_{i1} & y n_{i1} & 0 & 0 & n_{i2} \\ 0 & 0 & n_{i2} & x n_{i2} & n_{i1} \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \end{Bmatrix} = [R]_i \{\beta\}$$

from which

$$[R]_i^T [L] = \begin{bmatrix} n_{i1}(1 - a/l_i) & 0 & n_{i1}a/l_i & 0 \\ y n_{i1}(1 - a/l_i) & 0 & y n_{i1}a/l_i & 0 \\ 0 & n_{i2}(1 - a/l_i) & 0 & n_{i2}a/l_i \\ 0 & x n_{i2}(1 - a/l_i) & 0 & x n_{i2}a/l_i \\ n_{i2}(1 - a/l_i) & n_{i1}(1 - a/l_i) & n_{i2}a/l_i & n_{i1}a/l_i \end{bmatrix}$$

and $[G]_i = \int_{\partial v_i} [R]_i^T [L] ds$ for the side i .

The contribution to components $G_{I,J}$ for the side i is given by

J/I	$2i - 1$	$2i$	$2i + 1$	$2i + 2$
1	$n_{i1}l_i/2$	0	$n_{i1}l_i/2$	0
2	$n_{i1}l_i(y_i/3 + y_{i+1}/6)$	0	$n_{i1}l_i(y_i/6 + y_{i+1}/3)$	0
3	0	$n_{i2}l_i/2$	0	$n_{i2}l_i/2$
4	0	$n_{i2}l_i(x_i/3 + x_{i+1}/6)$	0	$n_{i2}l_i(x_i/6 + x_{i+1}/3)$
5	$n_{i2}l_i/2$	$n_{i1}l_i/2$	$n_{i2}l_i/2$	$n_{i1}l_i/2$

It is interesting to note that for a four node element if only three β -terms are used, the element stiffness matrix obtained by assumed stress hybrid method becomes exactly same as that obtained by Mallett's finite difference element scheme. Usage of three β -terms means assumption of constant σ_x , σ_y and τ_{xy} within the element which is also the case in finite difference element scheme.

3.2 Invariance of Stiffness Matrix

The stiffness matrix of an element calculated on the basis of global coordinates is not invariant as not all the assumed stresses are complete polynomials of the same degree. However, invariance may be achieved, as suggested by Cook [3], without changing the stress polynomial by calculating the element stiffness in a local coordinate system having a fixed orientation with respect to the element, regardless of how the element may be oriented in global coordinates. Thus if a given structure and its loads are rotated some amount in global coordinates, nodal loads associated with each element maintain their original direction with respect to the local element coordinates. Thus the element response becomes invariant as it is not affected by the rotation.

Once a local coordinate system is defined the calculation proceeds in the following manner.

1. Nodal coordinates of an element is transferred to the local system and the stiffness matrix is calculated in the local system.
2. Element stiffness matrix in the local system is then transformed to the global coordinate system and assembled in the global stiffness matrix. The transformation is accomplished by the operation

$$[k]_{global} = [A^T][k]_{local}[A] \quad (23)$$

where $[A]$ is the coordinate transformation matrix.

3. After assembly, element nodal displacements are calculated in the global xy directions.
4. To compute stresses, stress parameters β calculated in the local system are used. $\{\sigma\}_{local} = [P]_{local}\{\beta\}_{local}$ gives stresses in the local system which are then transformed to the global system.

Choice of finding the angle, the local system makes with global system, is not unique and alternative definitions of this angle are possible. However, finding a local coordinate system for a polygonal element is a problem as a polygon does not exhibit any preferred direction. In the present formulation, the local coordinate of an element is defined by two different methods.

In the first method, one of the axes of the local coordinate system is aligned with the longest edge of the element. In other words, the angle the longest edge makes with one of the global axes, is the angle by which the global axes are rotated to obtain the local coordinate axes for the particular element.

In the second method, an element is first mapped to a master 4, 8 or 12 node plane isoparametric element in s - t space. If the element has less than or equal to 4 nodes, it is mapped to a 4 node master element, if it has nodes between 5 and 8 it is mapped to an 8 node master element and so on. Wherever necessary, extra nodes are created in the element mid sides so that there is a one to one correspondence between the nodes of the actual and master element. Jacobian matrix for this transformation is then calculated and decomposed into

$$[J] = [R][U] \quad (24)$$

where $[R]$ represents a pure rotational transformation matrix and

$$[U] = \frac{1}{\sqrt{\text{trace}C + 2\sqrt{\det C}}}(C + \sqrt{\det C}I)$$

where

$$[C] = [J]^T[J]$$

From the above relationship $[R]$ is obtained as

$$[R] = [J][U]^{-1} \quad (25)$$

This rotational transformation matrix then is used for rotating the global axis to obtain the local coordinate axes. Rest of the calculation proceeds as outlined earlier.

3.3 Numerical Examples

Stretching Problem:

A square material body of size 40 mm x 40 mm is discretized into 24 voronoi polygons, as shown earlier in Fig. 1, and is subjected to a uniform tensile force of 100 KN. Analysis of this problem by the Assumed Stress Hybrid Method yields exact end deflection of 4.55 mm in X direction and σ_x of 2.5 kN/sq.mm in all elements. Also the displacement of the nodes in X direction versus their position from the fixed end has been found to be linearly proportional (Fig. 2). Thus both Mallett's method and assumed stress hybrid method yield exact results of a stretching problem.

The material body is now aligned at 45° to the global axes and is subjected to the same stretching load (Fig. 6). The results of deflection and stress in the direction the load are again found to be 4.55 mm and 2.5 KN/sq.mm which are exact figures.

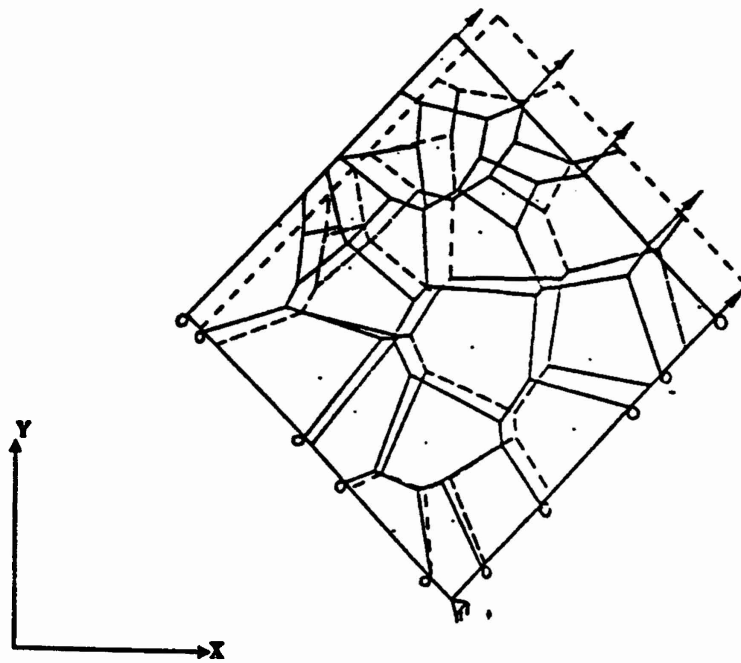


Fig. 6. Body oriented at 45° and subjected to tensile load.

Simple Shear Problem:

The same 40 mm x 40 mm material body discretised with the same 24 voronoi polygons is subjected to suitable displacement boundary condition so as to simulate simple shear (Fig. 7). Assumed stress hybrid method yielded exact value of $\tau_{xy} = 0.0385$ KN/sq.mm and $\sigma_x = \sigma_y = 0$ in every element.

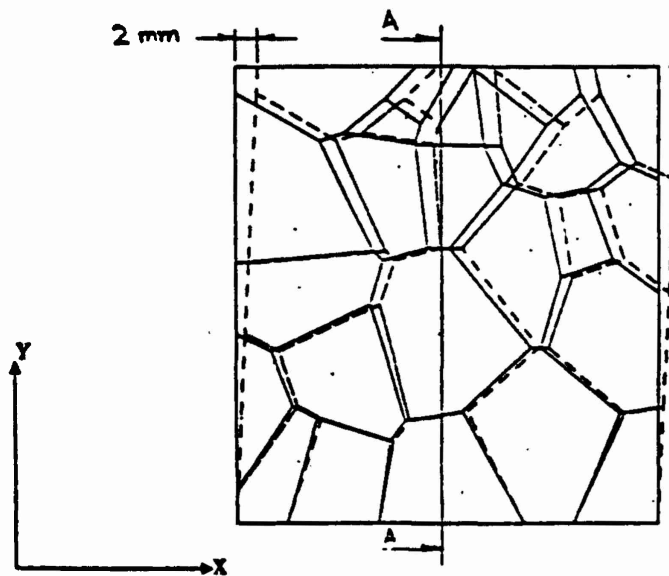


Fig. 7. Body subjected to simple shear.

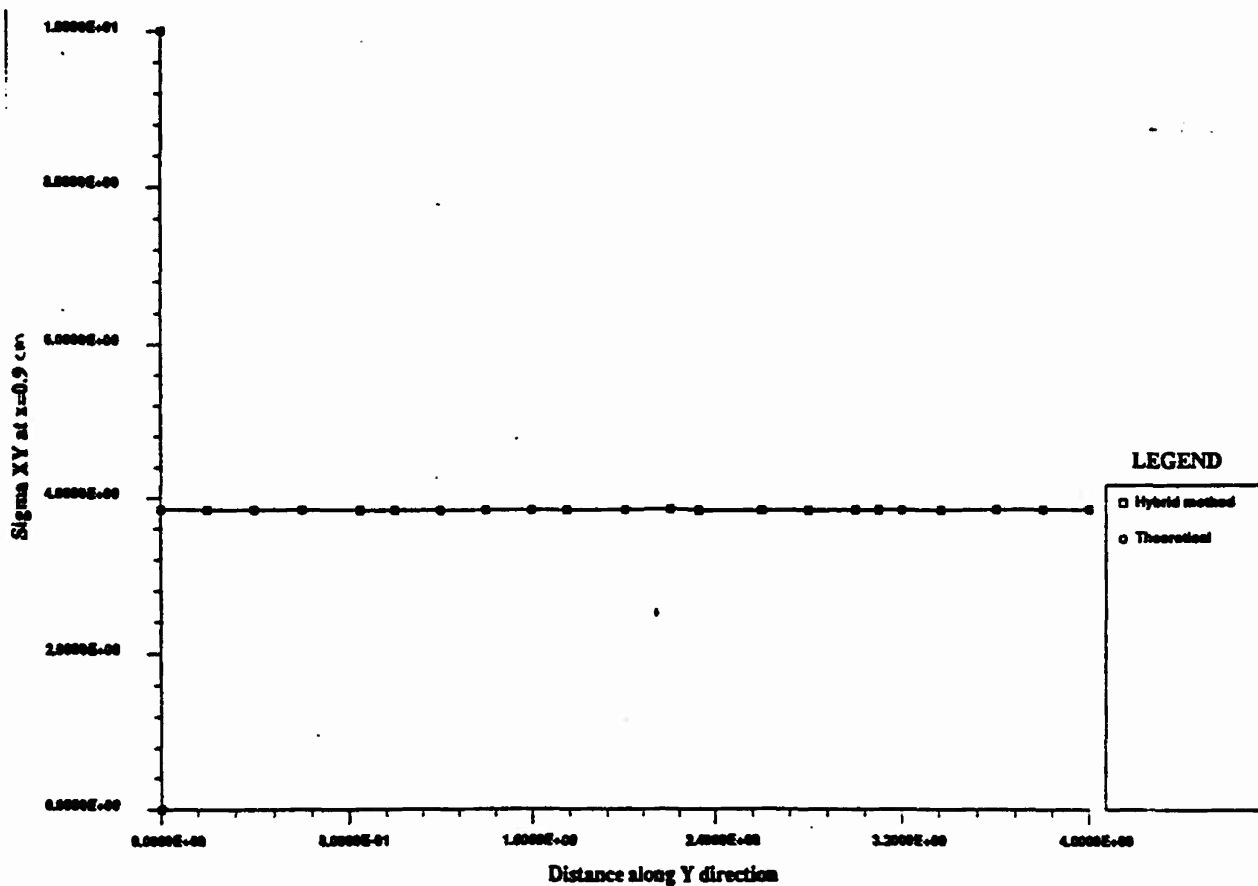


Fig. 8. Shear stress across section AA

Bending Problems:

Example 1.

A material body of size 40 mm x 160 mm is used as a cantilever beam with one end fixed and other end subjected to pure moment of 40 KN-mm. The beam is discretized into 28 voronoi polygon elements (Fig. 9). Longitudinal stress σ_x along a transverse section calculated by assumed stress hybrid method conformed more closely to the theoretical values than that obtained by Mallett's method (Fig. 10). The vertical displacements of the nodes at the top surface agree very well with the theoretical values (Fig. 11). The calculated tip deflection of 4.175 mm has an error of 4.4% with respect to the theoretical deflection of 4.368 mm.

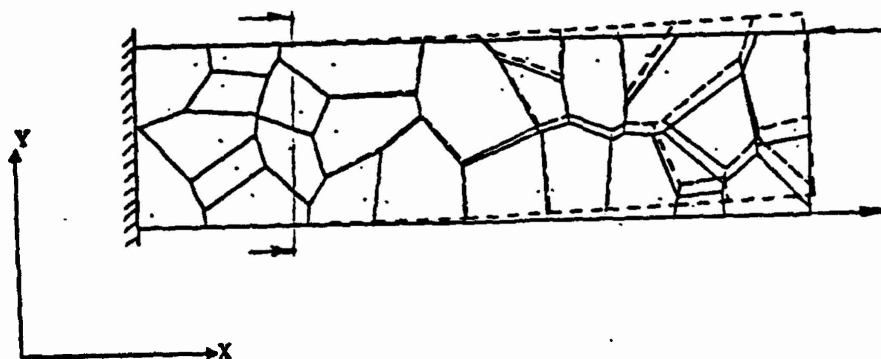


Fig. 9. Beam subjected to pure end moment (28 elements).

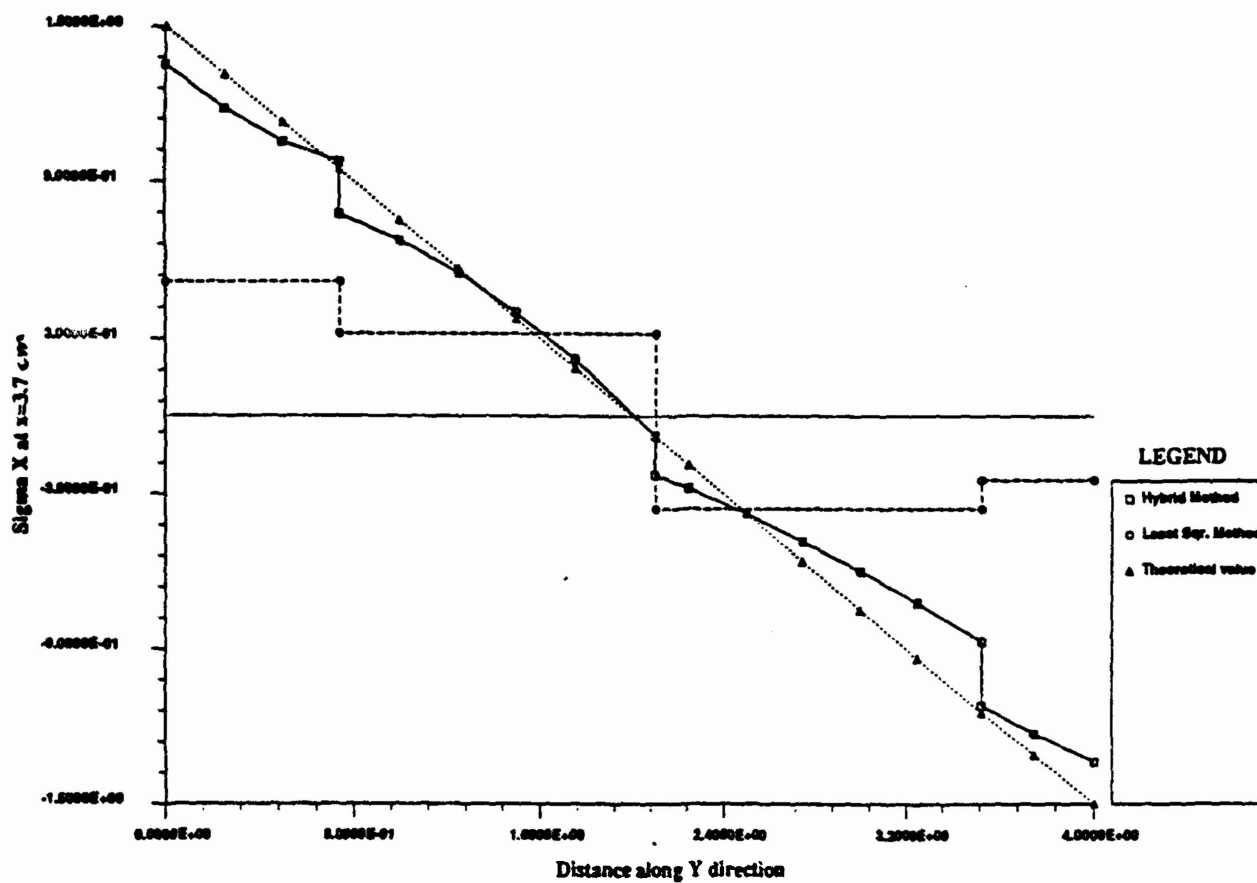


Fig. 10. Sigma X along a transverse section.

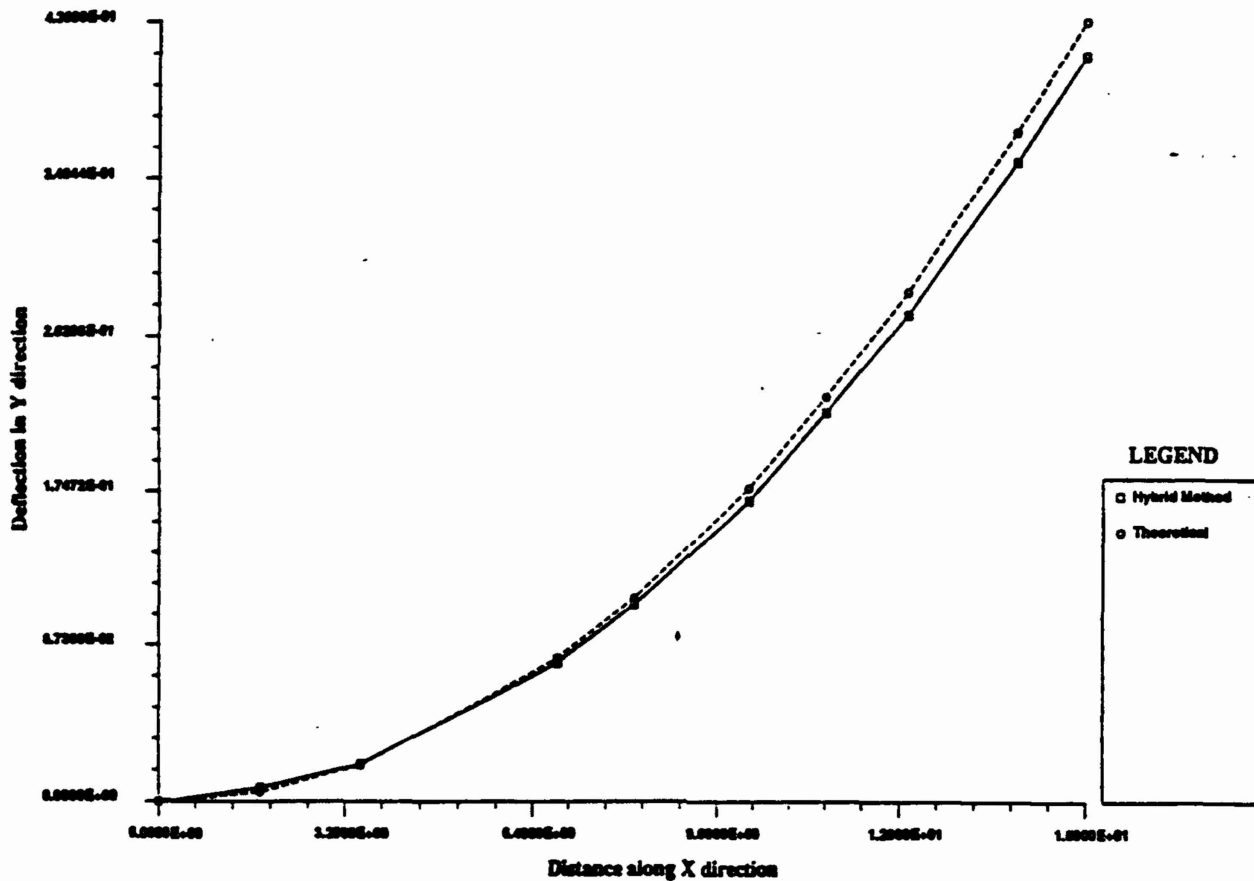


Fig. 11. Deflection of top surface.

Examples 2 and 3.

The same 40 mm x 160 mm beam is now discretized into 53 and 152 voronoi polygon elements respectively (Fig. 12 & 13) and subjected to pure end moment of 40 KN-mm. σ_x along a transverse section is plotted (Fig. 14) for all three cases, i.e. beam discretized by 28 elements, 53 elements and 152 elements. The calculated tip deflection for the beam with 53 elements is 4.346 mm, an error of 0.5% compared to the theoretical deflection. The corresponding figures for the beam with 152 elements are 4.452 mm and 1.9% respectively.

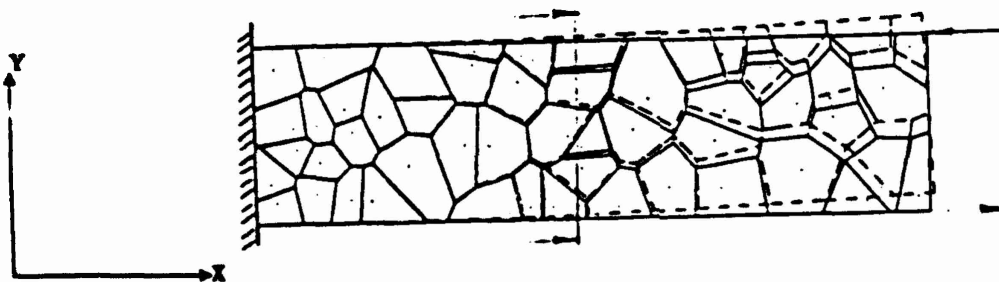


Fig. 12. Beam subjected to pure end moment (53 elements).

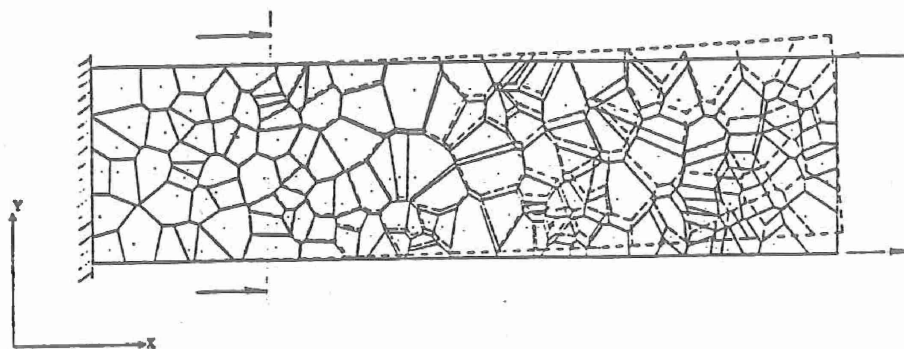


Fig. 13. Beam subjected to pure end moment (152 elements).

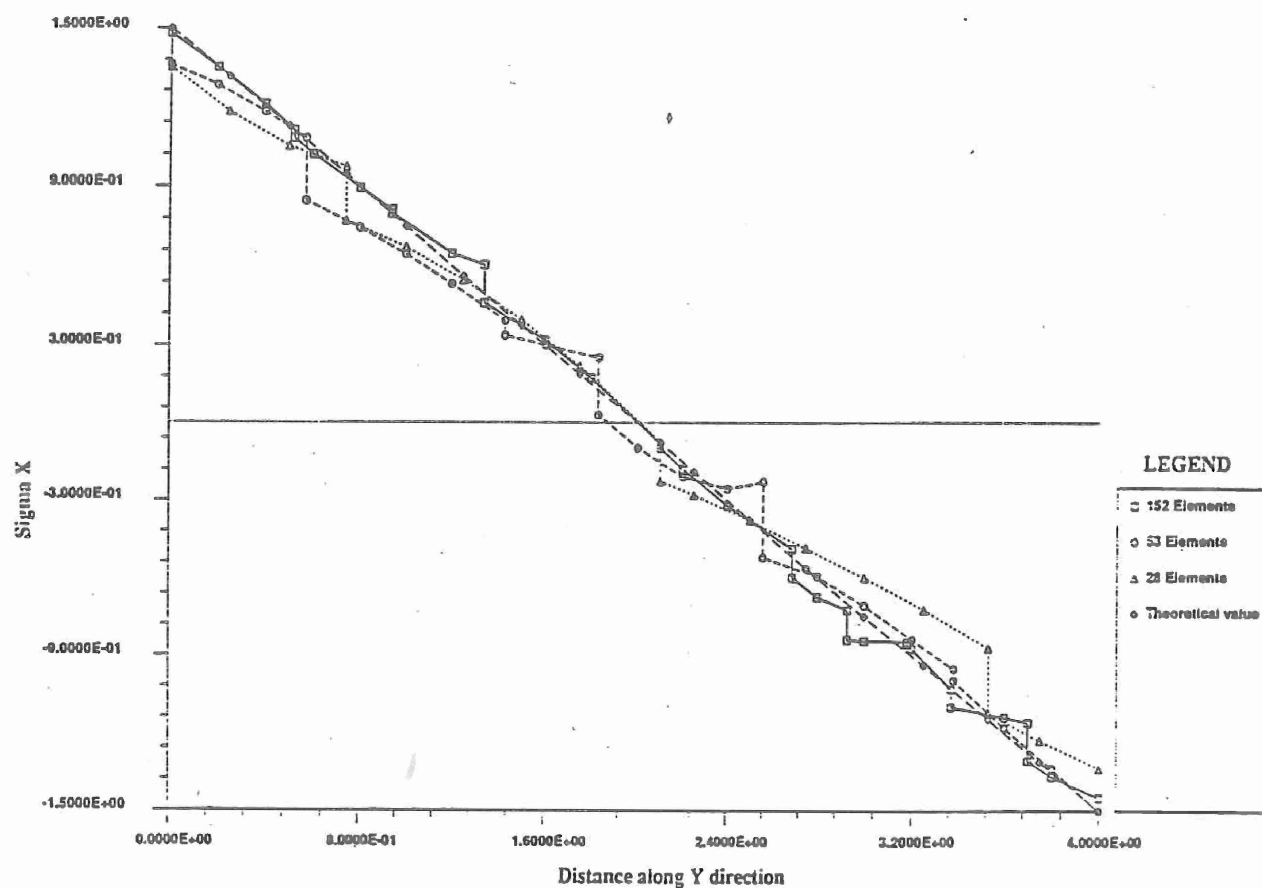


Fig. 14. Sigma X along a transverse section.

Fig. 14 indicates that as more and more elements are used to discretize the beam, the stress distribution converges towards the theoretical value. Fig. 15 shows the distribution of σ_x throughout the beam discretized by 152 elements. Except near the area of application of moment load, where the Saint Venant effect is predominant, the σ_x stress distribution conforms well with the theoretical stress distribution.

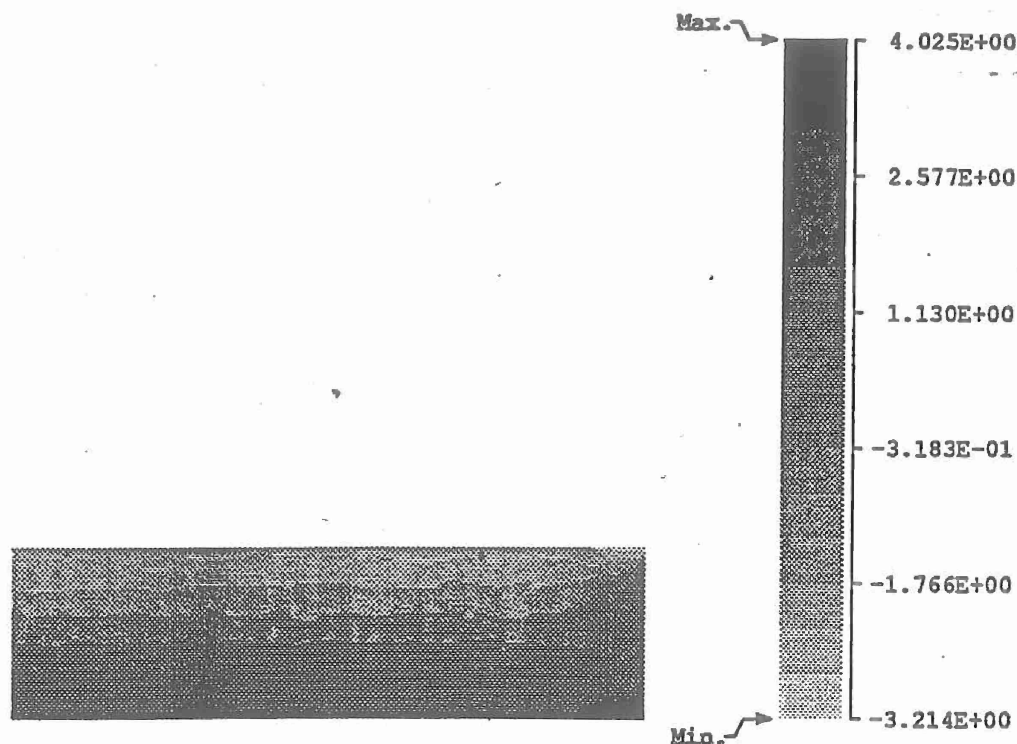


Fig. 15. Distribution of Sigma X in beam having 152 elements.

The beam, discretized into 53 elements, is now aligned at 45° with the global axis (Fig. 16). Stresses in some elements deteriorate if the calculations are based on global coordinate. However, use of a local coordinate system for every element improves the results and makes the elements invariant with respect to the orientation of the beam with the global axis (Fig. 17). Local coordinate for an element is usually obtained by rotating the global system until one of the axes aligned with the largest edge of the element, and is identified as Cook's method. In this example the matrix used to transform the global system to the local coordinate system is also obtained by decomposing the Jacobian matrix, as detailed earlier, and is identified as Jacobian method. Results based on local coordinate system, obtained by either method, compare well with each other.

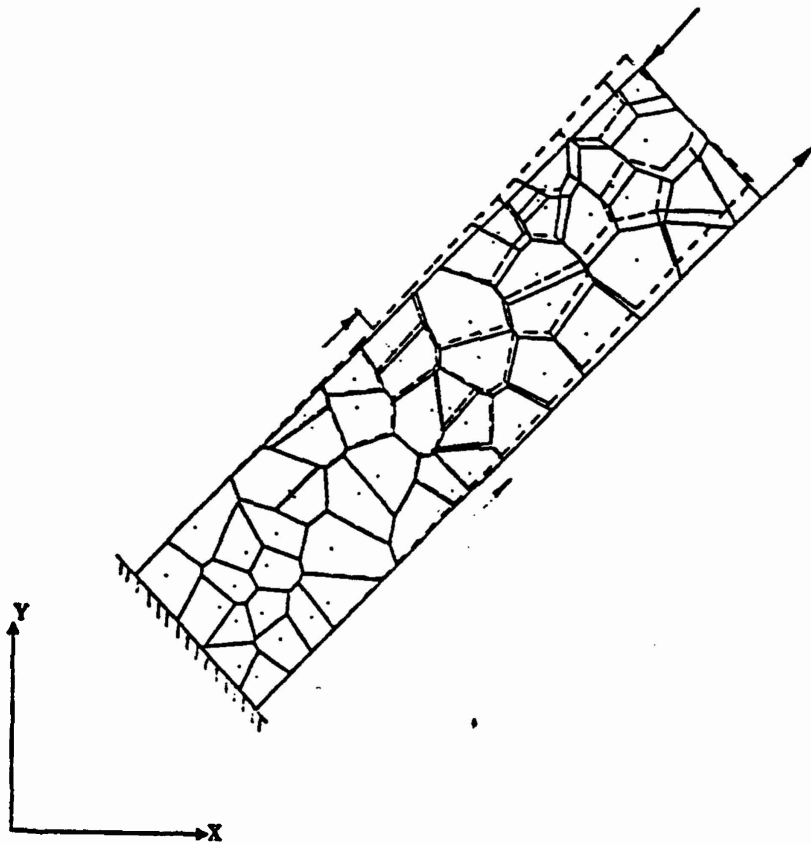


Fig. 16. Beam oriented at 45° and subjected to end moment(53 elements).

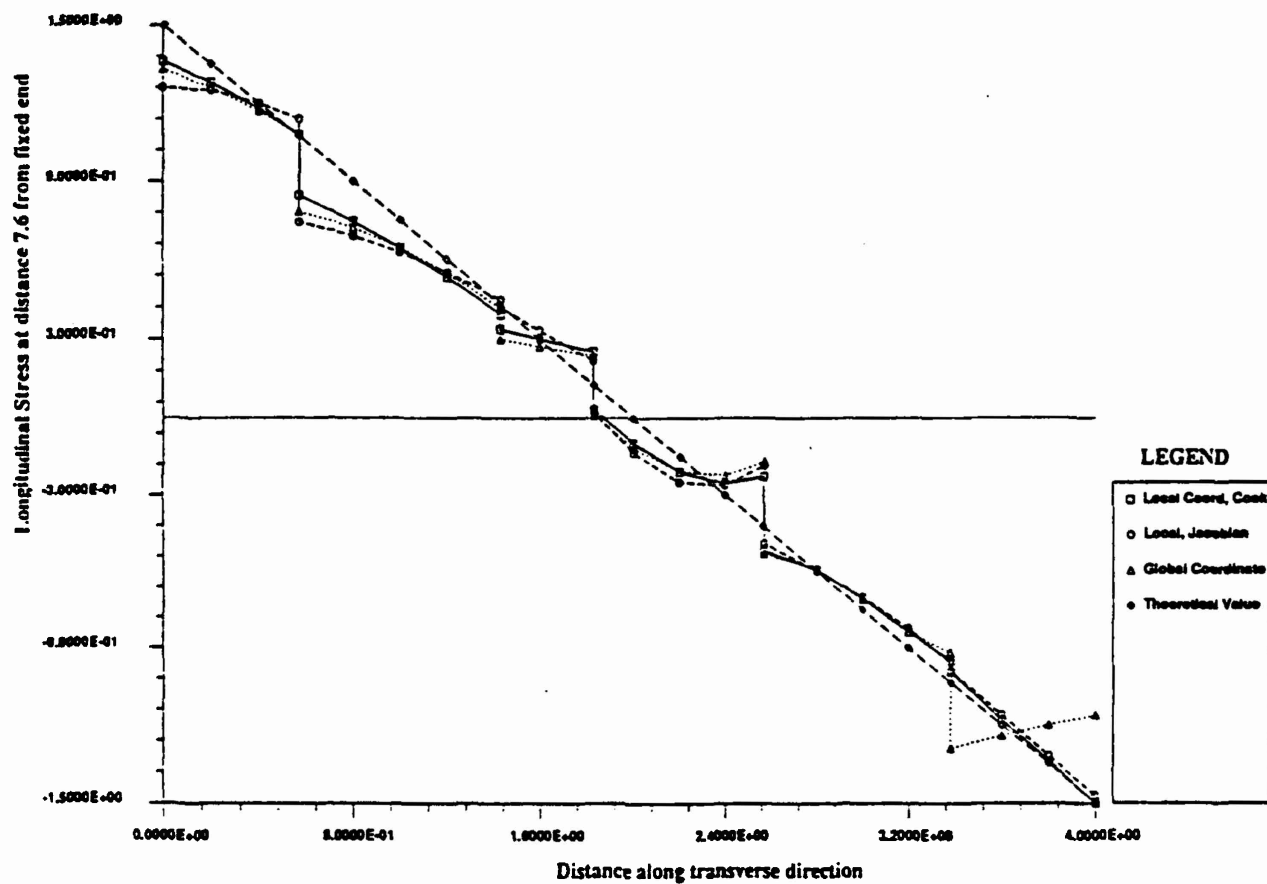


Fig. 17. Longitudinal stress along a transverse section using Local & Global coord.

The above exercise is carried out on the beam with 152 elements, aligned at 45° with the global axis. Results based on local coordinate system are found to improve (Fig. 18).

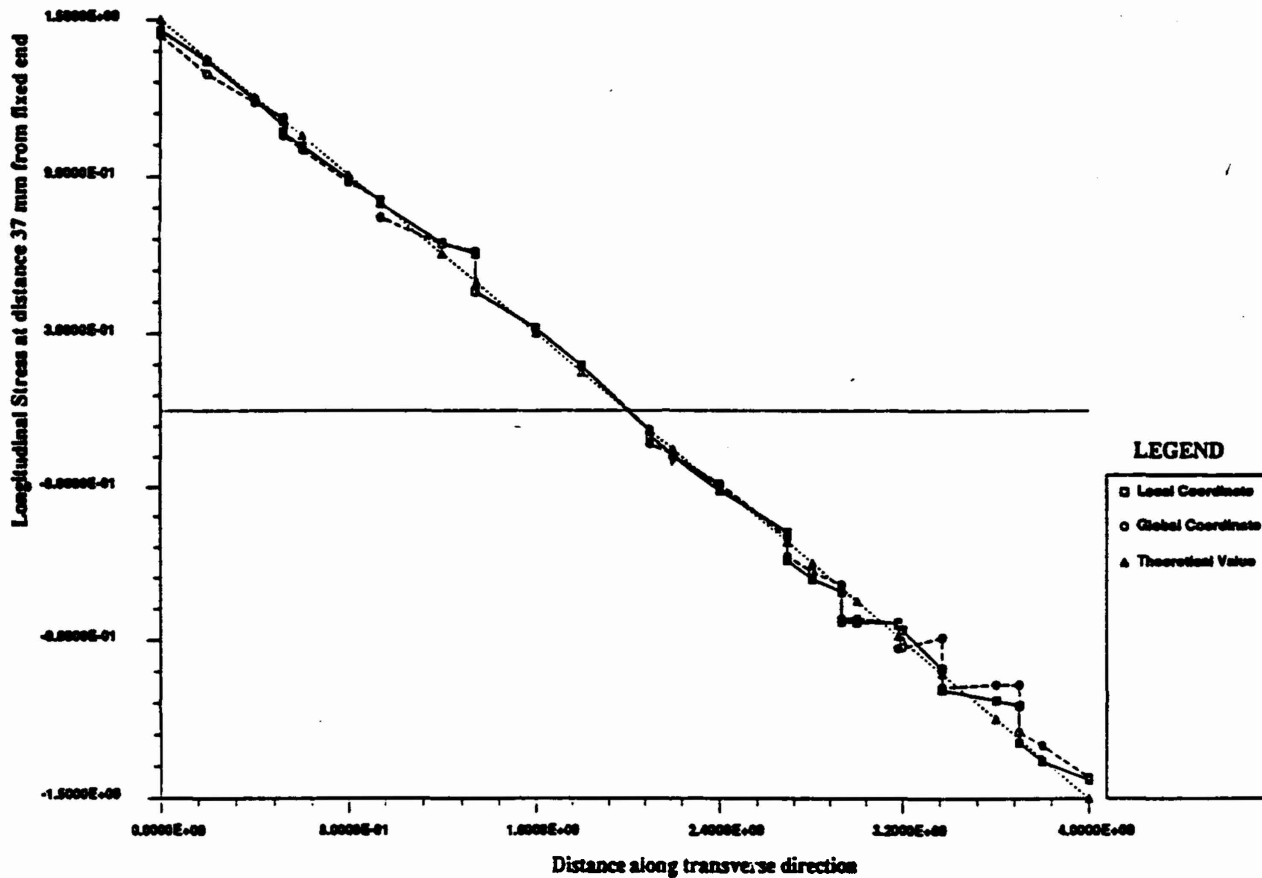


Fig. 18. Longitudinal stress along a transverse section using Local & Global coordinate. (152 elements)

Example 4.

Having shown that the results improve as the number of elements increase, the same 40 mm x 160 mm beam is discretized into 86 elements (Fig. 19), most of which are of regular hexagonal shape, and the same exercise is carried out. Fig. 20 indicates that longitudinal stresses in the elements across a transverse section agree very well with the theoretical values. Fig. 21 shows the distribution of σ_x in the entire beam. Except near the area of application of moment load, where the Saint Venant effect is predominant, the σ_x stress distribution conforms very well with the theoretical stress distribution.

Results of this example indicate that as the elements become more regular in shape, the results improve.

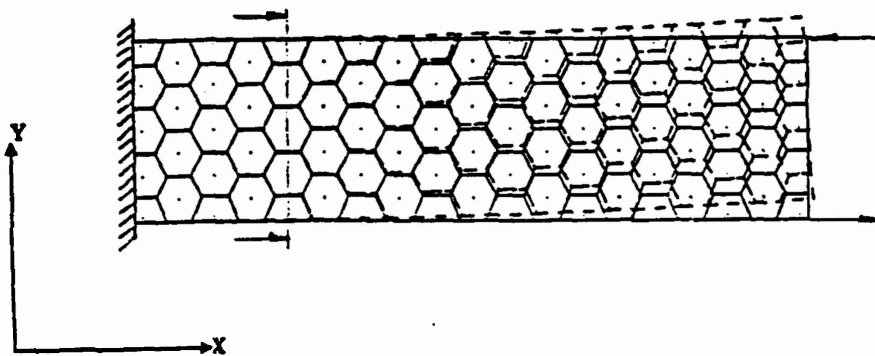


Fig. 19. Beam subjected to pure end moment (86 elements).

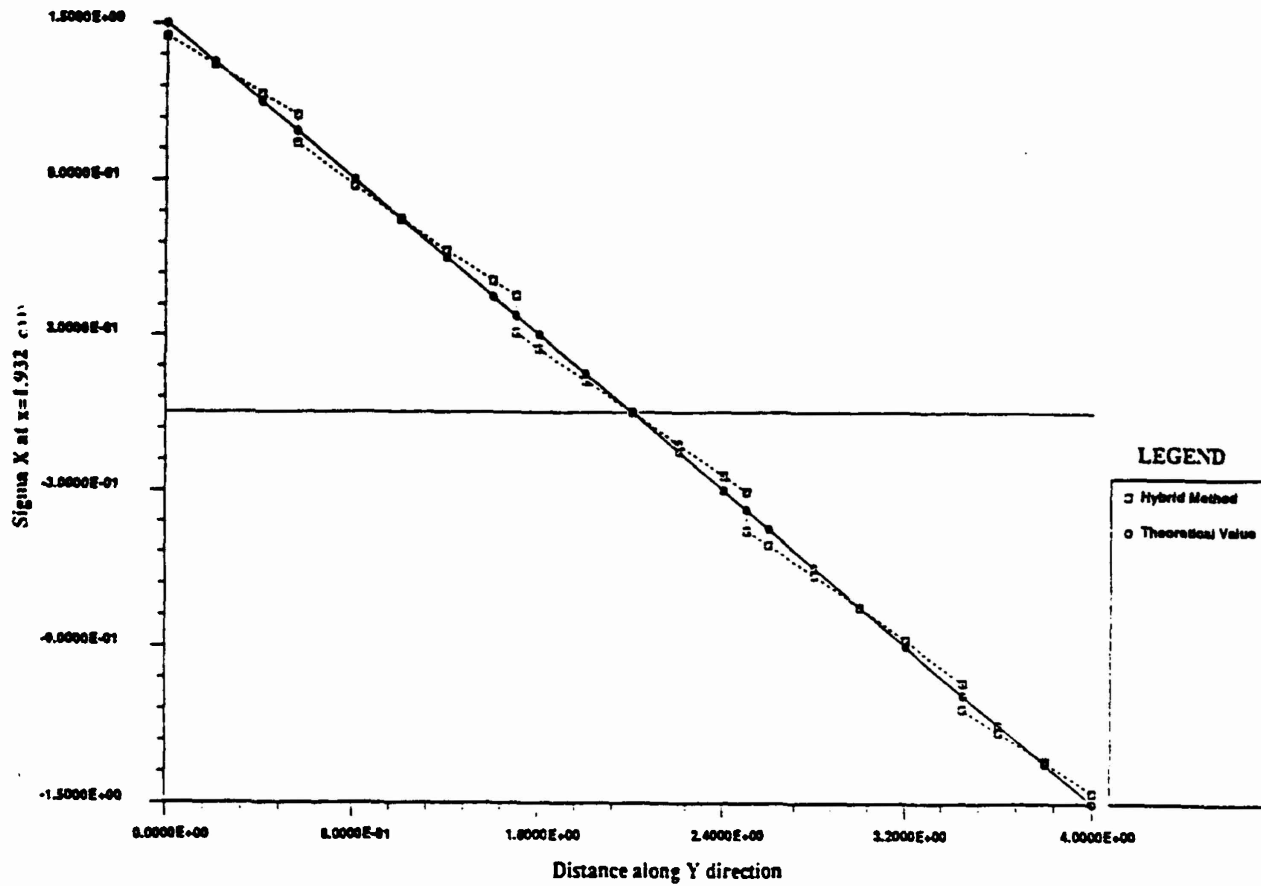


Fig. 20. Sigma X along a transverse section (86 elements).

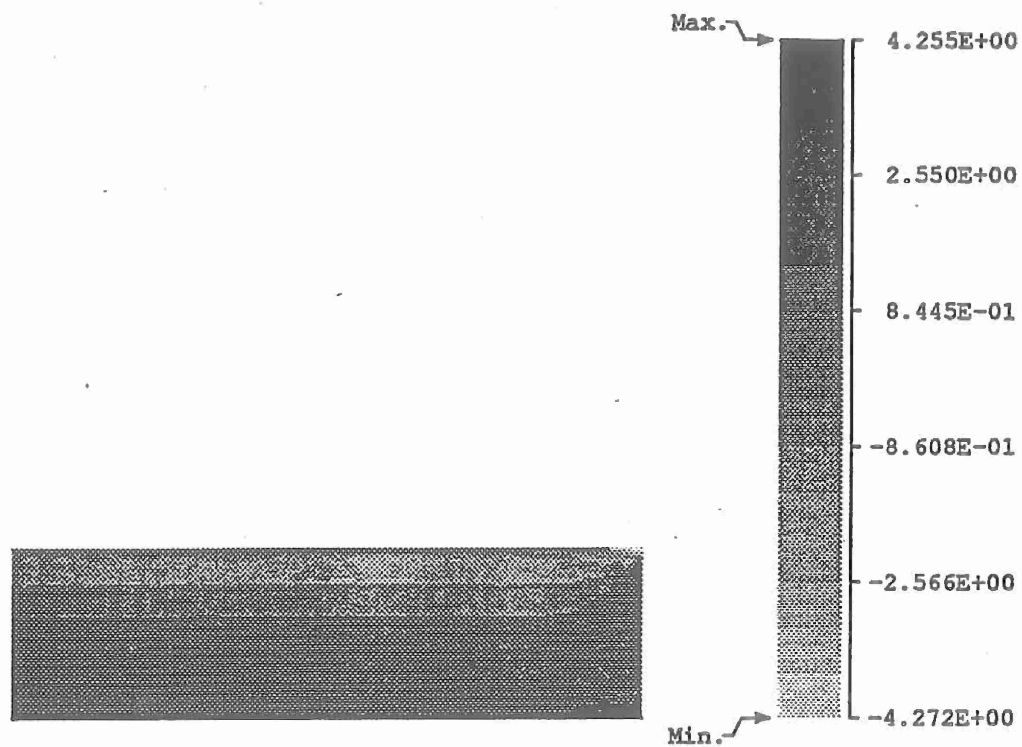


Fig. 21. Distribution of Sigma X in beam having 86 elements.

The beam is now oriented at 45° with the global axis (Fig. 22). Fig. 23 indicates that a local coordinate system for every element produces better results than that obtained by a global coordinate system.

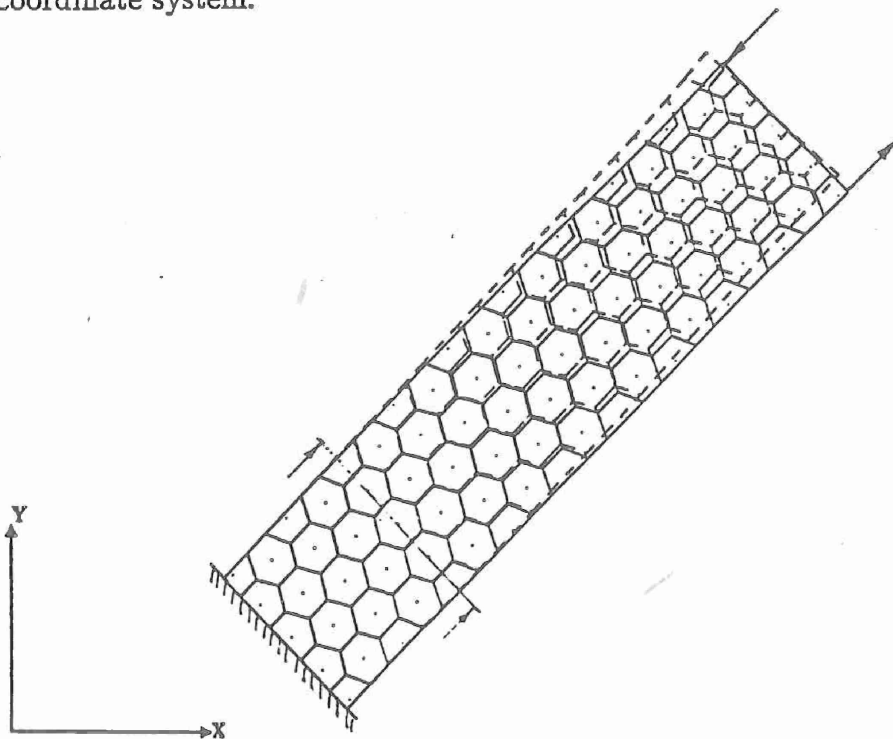


Fig. 22. Beam oriented at 45° and subjected to end moment.

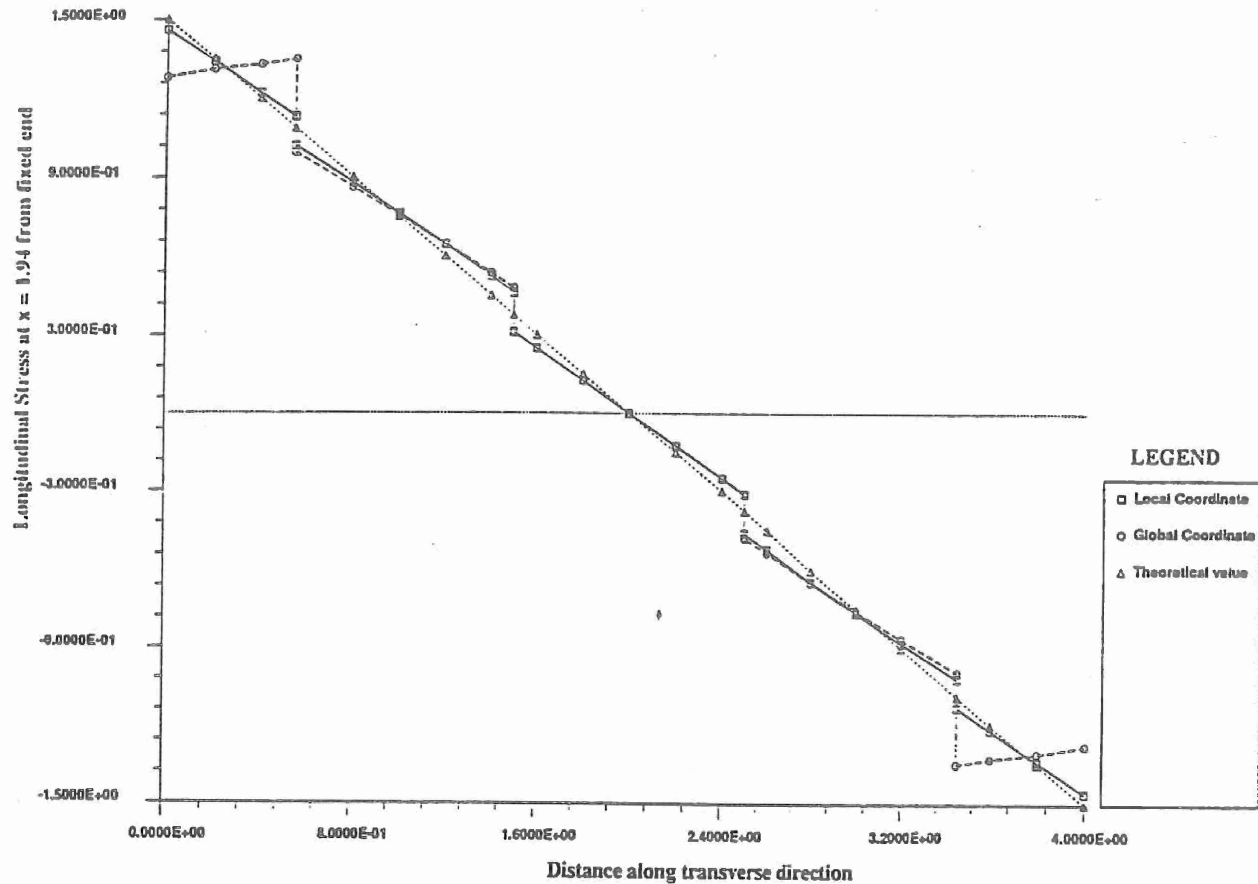


Fig. 23. Longitudinal stress along a transverse section using Local & Global coord.

Example 5.

The material body of size 40 mm x 160 mm, with regular 86 elements, is used as a cantilever beam subjected to a downward acting load of 0.5 kN at the end (Fig. 24). The load is distributed over the edge in a parabolic distribution such that the total area under the parabola is equal to the load 0.5 kN. Shear stress across a transverse section is plotted in Fig. 25 and found to agree well with the theoretical distribution.

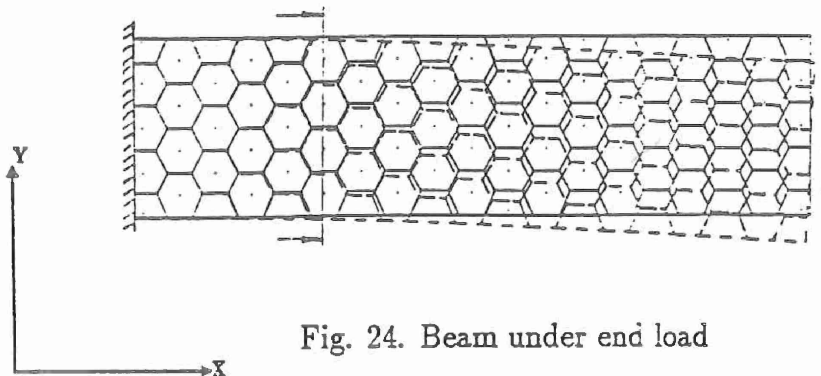


Fig. 24. Beam under end load

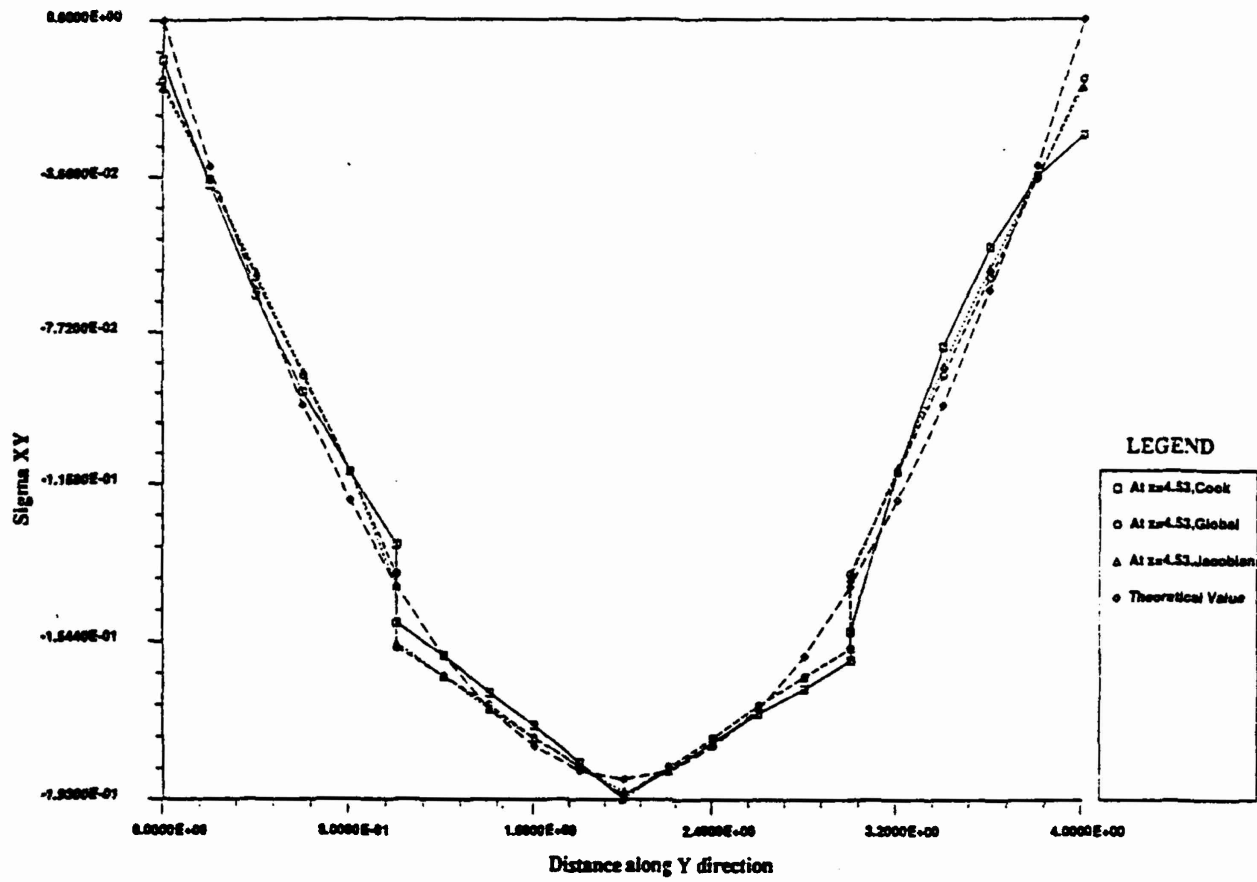


Fig. 25. Shear stress distribution along a transverse section.

4 Modelling of Second Phase Material Present in Voronoi Cell

Having established a finite element formulation for polygonal elements with isotropic material properties, next step is to account for the second phase particulate present in every element. The problem of modelling the presence of a second phase material within a finite element was addressed by Accorsi [4] by introducing a transformation strain in those regions. The finite element method reflects the shape, size and location of the second phase material within the element.

4.1 Problem Formulation

The formulation is based on dividing the total problem into two separate subproblems. First, called homogeneous problem, corresponds to the finite element analysis of the body without material discontinuity and the second, called the deviation problem is defined by the difference between the actual and homogeneous field quantities. The solution of the actual problem can be found by adding the corresponding solutions of homogenous and deviation problems.

Let superscripts t , o and $'$ indicate the actual or nonhomogenous problem, the homogenous problem and the deviation problem respectively.

The constitutive relation for the actual material is thus given as

$$\sigma^t = C(x)\epsilon^t \quad (26)$$

where $C(x)$ is the elasticity matrix.

The transformation strain mentioned earlier is defined by the following equation.

$$\sigma^t = C(x)\epsilon^t = C^o(\epsilon^t - \epsilon^*) \quad (27)$$

where C^o is the elasticity matrix of homogenous material and ϵ^* is the strain in the discontinuity. From the above relationship one obtains

$$\epsilon^t = [C^o - C(x)]^{-1} C^o \epsilon^* = H(x) C^o \epsilon^* \quad (28)$$

where $H(x) = [C^o - C(x)]^{-1}$

Now as per earlier definition

$$\epsilon^t = \epsilon^o + \epsilon' \quad (29)$$

or

$$\epsilon^o + \epsilon' = H(x) C^o \epsilon^* \quad (30)$$

The constitutive relation for the deviation problem is given as

$$\sigma' = \sigma^t - \sigma^o = C^o(\epsilon' - \epsilon^*) \quad (31)$$

Discretization of the homogenous problem is straight forward. Using the principle of minimum potential energy, the element equations are obtained. The assembled equations for the homogenous problem are written as

$$K^{\circ} \delta^{\circ} = F^{\circ} \quad (32)$$

where $K^{\circ}, \delta^{\circ}, F^{\circ}$ are the global stiffness matrix, global displacement vector and global load vector respectively. For the purpose of solving the above equation, they can be rewritten as

$$\delta^{\circ} = (K_m^{\circ})^{-1} F_m^{\circ} \quad (33)$$

where K_m°, F_m° indicate K°, F° modified for displacement boundary conditions. Similarly, the element equation for the discretized deviation problem is written as

$$K_e^{\circ} \delta_e' = F_e^{\circ} \quad (34)$$

where

$$F_e^{\circ} = \int_{v_e} B^T C_e^{\circ} \epsilon^{\circ} dv \quad (35)$$

B is the strain displacement matrix such that

$$\epsilon_e' = B \delta_e' \quad (36)$$

and K_e° is the elemental stiffness matrix.

For an element of matrix material C_e° containing m_e microstructural discontinuities with material property $C^r, r = 1, 2, \dots, m_e$ and occupying regions Ω_r , the transformation strain is non zero only within the region Ω_r . The equation (35) becomes

$$F_e^{\circ} = \sum_{r=1}^{m_e} \int_{\Omega_r} B^T C_e^{\circ} \epsilon^{\circ} dv \quad (37)$$

As the transformation strain ϵ° is constant within a discontinuity the above equation (37) is further reduced to

$$F_e^{\circ} = \sum_{r=1}^{m_e} (R_e^r)^T C_e^{\circ} \epsilon^{\circ r} \quad (38)$$

where

$$R_e^r = \int_{\Omega_r} B dv$$

Assembly of the equation (34) gives

$$K^{\circ} \delta' = F^{\circ} \quad (39)$$

where

$$F^* = \sum_e \phi_e^T F_e^*$$

ϕ_e is a transformation matrix which when operated on global displacement vector δ returns the element displacement vector δ_e .

Equation (39), when modified for displacement boundary condition, takes the form

$$K_m^o \delta' = \psi F^* \quad (40)$$

Since the displacement boundary conditions for the homogenous and deviation problems correspond to the same nodes, the modification to the stiffness matrix is same for both the problems. For the deviation problem, wherever the displacement boundary conditions are zero, zeros are placed in the corresponding positions in F^* . To carry out this modification of F^* a matrix ψ is used. ψ is an identity matrix with zeros in the diagonal in the positions corresponding to the displacement boundary conditions.

From equation (40)

$$\delta' = (K_m^o)^{-1} \psi F^* \quad (41)$$

Therefore

$$\begin{aligned} \delta'_e &= \phi_e \delta' \\ &= \phi_e (K_m^o)^{-1} \psi F^* \\ &= \phi_e (K_m^o)^{-1} \psi \sum_e \phi_e^T \sum_{s=1}^{m_e} (R_e^s)^T C_e^o \epsilon^{*s} \\ &= \phi_e (K_m^o)^{-1} \psi \sum_{s=1}^M \phi_e^T (R_e^s)^T C_e^o \epsilon^{*s} \end{aligned} \quad (42)$$

where M is the total number of discontinuities in the system.

Writing equation (30) in terms of nodal displacements for an element, one obtains

$$B \delta_e^o + B \delta'_e = H(x) C_e^o \epsilon^* \quad (43)$$

Integrating equation (43) over Ω_r , the domain of r th discontinuity in the element, one obtains

$$R_e^r \delta_e^o + R_e^r \delta'_e = V_r H_e^r C_e^o \epsilon^{*r}$$

where V_r is volume of r th discontinuity and $H_e^r = [C_e^o - C^r]^{-1}$.

Rearranging the above equation

$$V_r H_e^r C_e^o \epsilon^{*r} - R_e^r \delta'_e = R_e^r \delta_e^o = R_e^r \phi_e^r \delta^o \quad (44)$$

Substituting the value of δ'_e from equation (42) into equation (44) we have

$$\sum_{s=1}^M [V_r H_e^s \delta^{rs} - R_e^r \phi_e^r (K_m^o)^{-1} \psi \phi_e^{sT} R_e^{sT}] C_e^o \epsilon^{*s} = R_e^r \phi_e^r \delta^o \quad (45)$$

where δ^{rs} is the Kronecker delta. The above equation is now assembled by evaluating the terms for each discontinuity in the system $r, s = 1, 2, \dots, M$. The assembled equation takes the form

$$S(C^o \epsilon^*) = R \delta^o \quad (46)$$

It may be observed from equation (45) that ϵ^{*r} is function of R_e^r and H_e^r . $R_e^r (= \int_{\Omega_r} B dv)$ in turn is dependent upon the shape and size of the discontinuity, as the integration is over Ω_r , and also on the location of the discontinuity within the element due to the presence of B matrix in the expression. If the element to start with is not constant strain type, B is a function of the coordinates within the element. H_e^r matrix contains the material property of the discontinuity. Thus ϵ^{*r} depends upon the shape, size and location of the discontinuity within the element.

The subroutine for calculation of δ' follows the following steps

- From equation (46) calculate $C_e^o \epsilon^{*r}$
- Calculate $F_e^* = \sum_{r=1}^{m_e} (R_e^r)^T C_e^o \epsilon^{*r}$ by substituting the value of $C_e^o \epsilon^{*r}$
- Find global $F^* = \sum_e \phi_e^T F_e^*$
- Substitute F^* in equation (41), to get $\delta' = (K_m^o)^{-1} \psi F^*$

A standard 2-D finite element program FEM-2 [5] is used to calculate δ^o and δ' , i.e. the solution of the homogenous and deviation problem respectively. Solution of the actual problem is thus $\delta^t = \delta^o + \delta'$.

4.2 Numerical Examples

Stretching Problem:

The homogenization formulation allows the discontinuity to be treated as a void by setting its material properties to zero. A square material body with a central hole is subjected to tensile load. Resulting stress distribution along a transverse section passing through the hole is calculated by conventional FEM and FEM coupled with homogenization method. For the first case (Fig. 26) material around the hole is discretized by four node elements and for the second case (Fig. 27), the material is discretized in such a way that the entire hole is contained within an element.

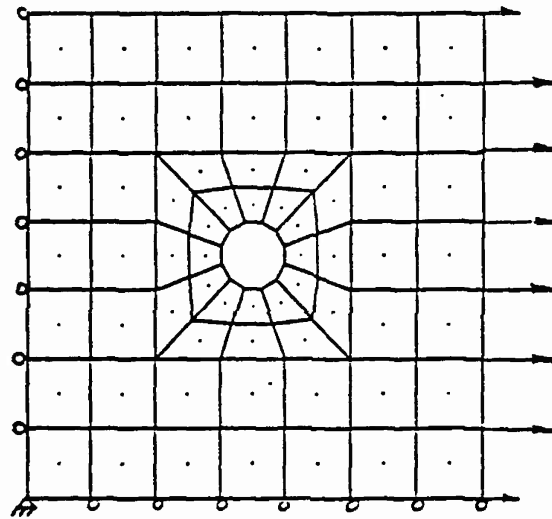


Fig. 26. Analysed by FEM2

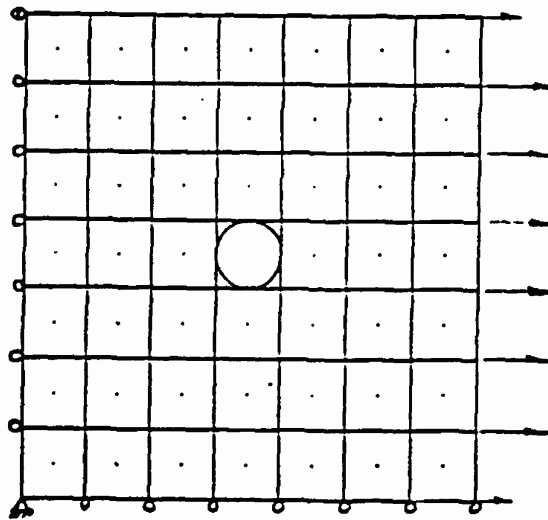


Fig. 27. Analysed by FEM2 with Homogenization.

Stress distribution (Fig. 28) by either method compare well up to a distance $\frac{D}{4}$ from the edge of the hole, D being the hole diameter. After that, the sharp rise in stress as the edge is approached is not reflected properly by homogenization method. This however, is expected as the homogenization method models the presence of the entire hole in an element in an average sense.

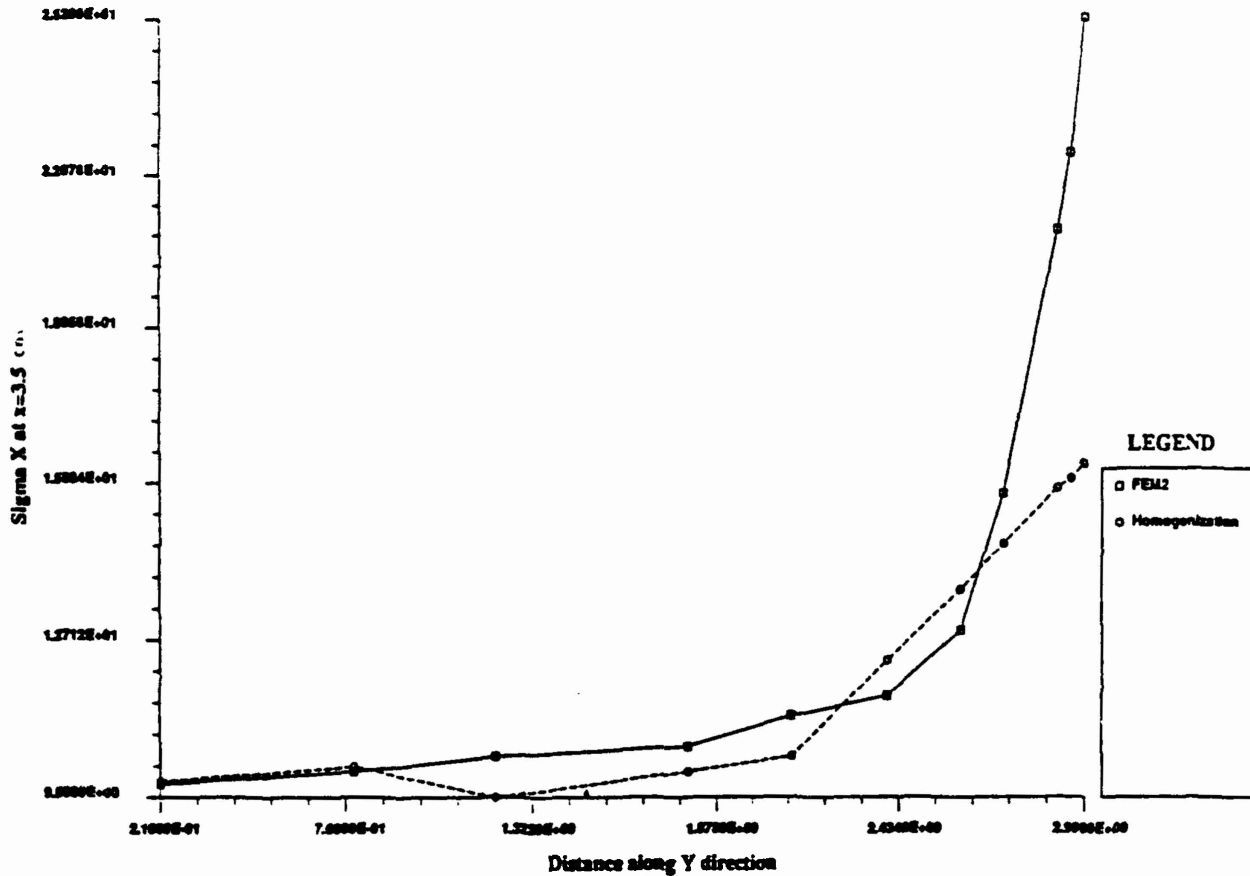


Fig. 28. Sigma X across a section through the hole.

Bending Problem:

A cantilever beam, having three second phase materials and subjected to a pure moment at the end is analyzed by conventional FEM and FEM coupled with homogenization method. For the conventional FEM the beam is discretized as shown in Fig. 29 and for the homogenization method the discretization is shown in Fig. 30. The Young's modulus of the second phase material is assumed as 4 KN/sq.mm which is double that of matrix material. Poisson's ratio is assumed as 0.3.

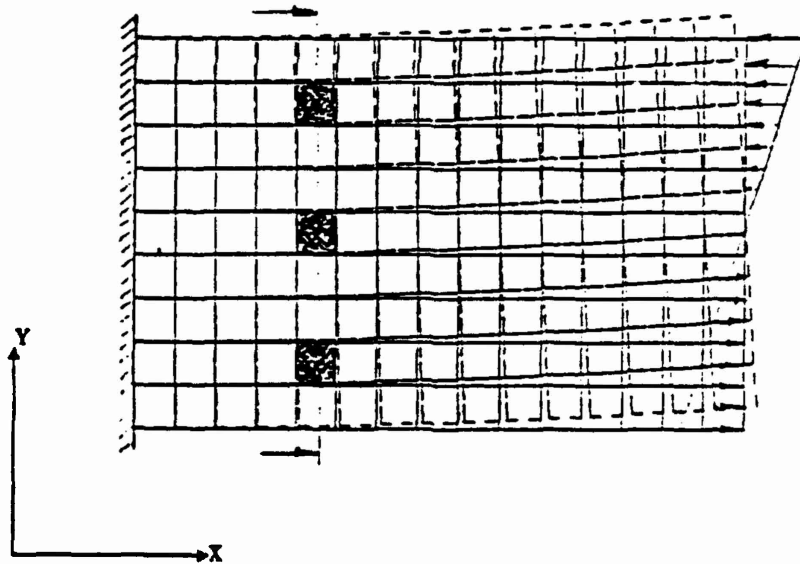


Fig. 29. Analysed by FEMC

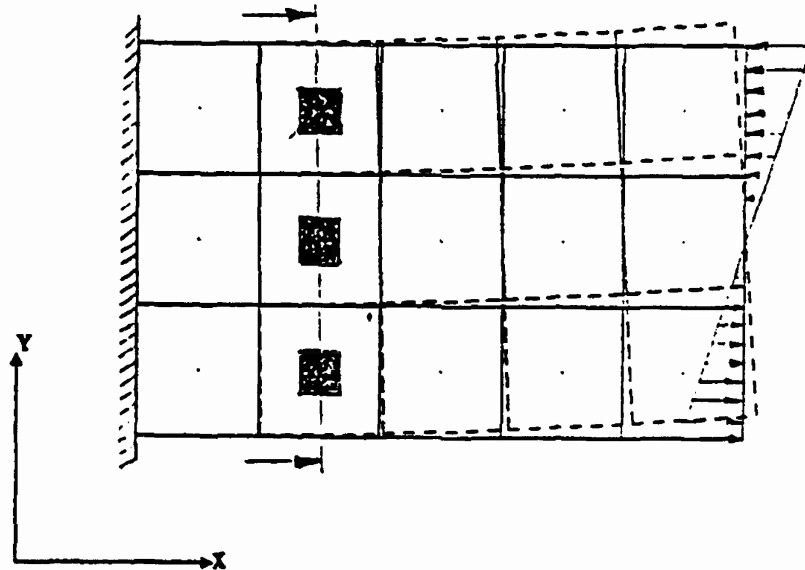


Fig. 30. Analysed by FEM2 with Homogenization.

The stress distribution, along a transverse section passing through the second phase materials, obtained by either method is plotted in Fig. 31. The stress predicted by homogenization technique is found to compare somewhat well in the matrix material, however within the second phase material, the stress is well above than that predicted by conventional FEM.

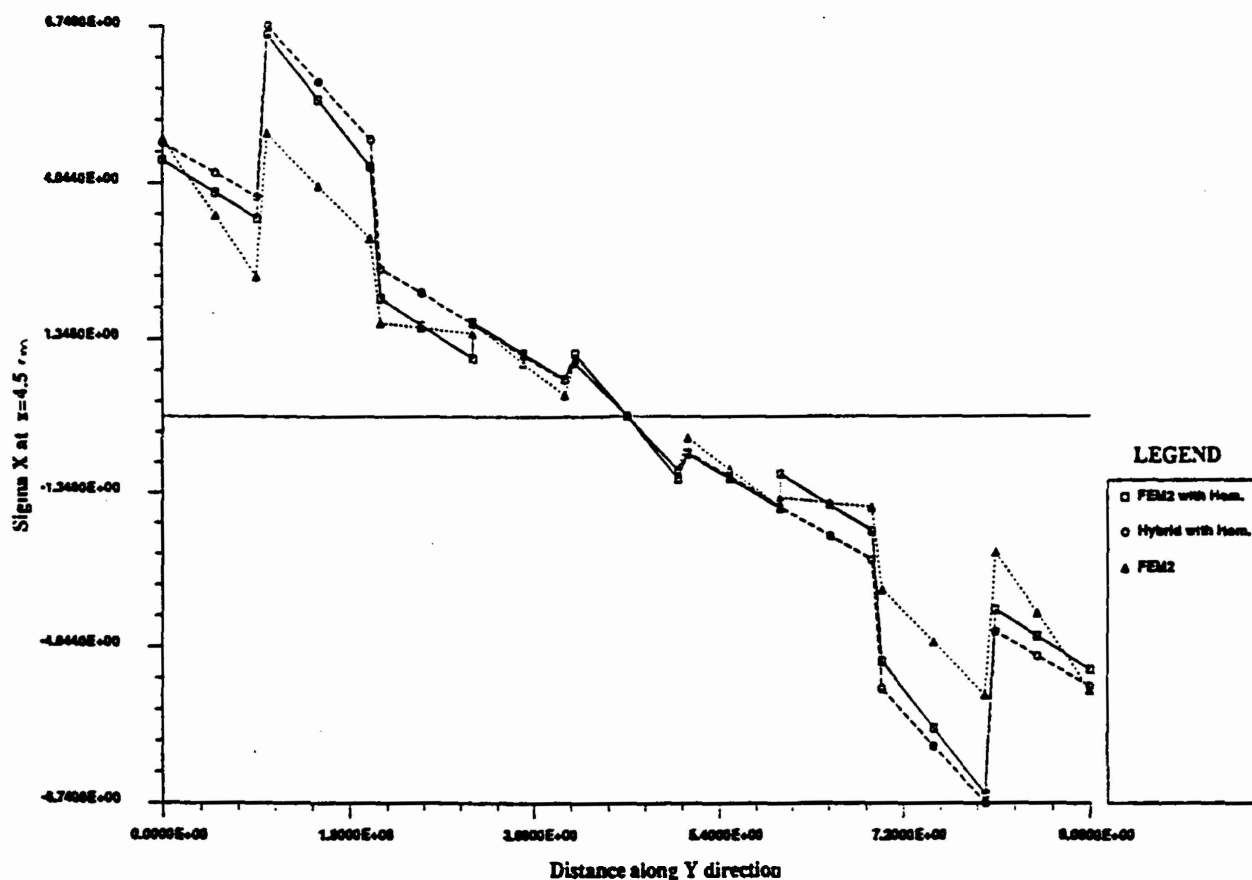


Fig. 31. Sigma X across a section through the second phase material.

All the numerical results of conventional FEM and homogenization methods, discussed so far, are based on a program for analysis of linearly elastic structures in a plane using 4-node isoparametric elements, called FEM2. However, for the analysis of composites with randomly dispersed second phase materials, the elements that will be encountered, as explained earlier in the report, will have more than four nodes. For the analysis of these polygonal elements, assumed stress hybrid method is introduced. The present problem of a beam dispersed with three second phase materials and subjected to a pure end moment is then analysed by the homogenization technique coupled with assumed stress hybrid method. The results, as shown in Fig. 30 compare very well with that obtained by homogenization method coupled with FEM2.

The same beam bending problem is solved by discretizing the beam into irregular polygonal elements (Fig. 32) and using homogenization coupled with assumed stress hybrid method. Longitudinal stresses along the same transverse section using irregular and regular mesh are plotted in Fig. 33. The results are found to compare well.

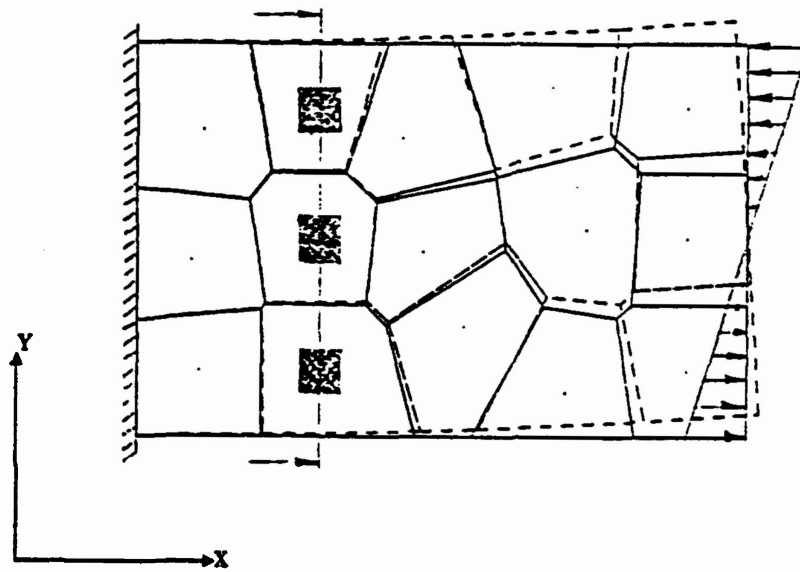


Fig. 32. Beam discretized by polygonal elements.

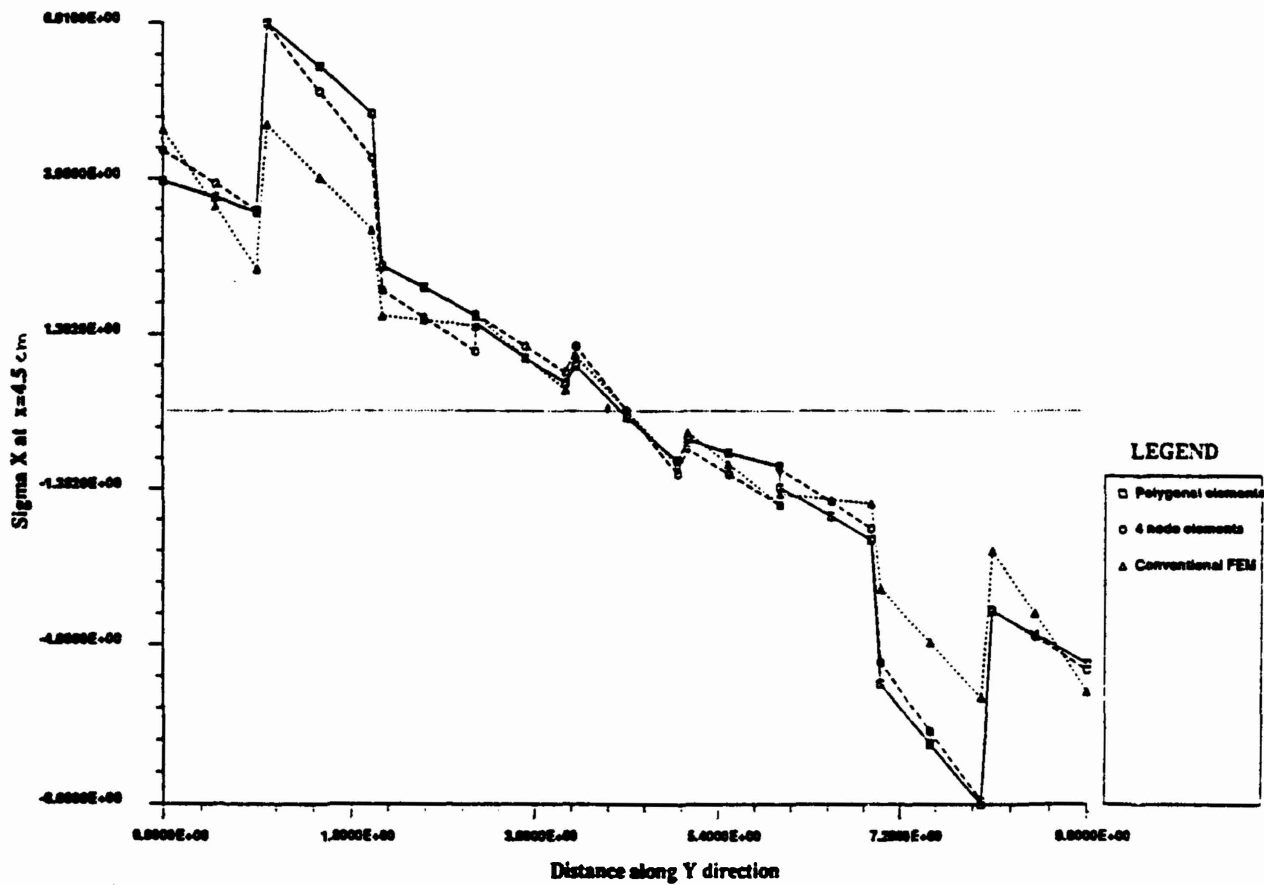


Fig. 33. Sigma X across a section through the second phase material.

Finally the beam of size 40 mmx 160 mm which was used in several earlier examples, is analysed by homogenization coupled with assumed stress hybrid method. The beam,

however, is now dispersed with 64 second phase particulates and is subjected to a pure end moment of 40 KN-mm(Fig. 34). Longitudinal stresses along a transverse section which passes through several second phase particulates are plotted in Fig. 35.

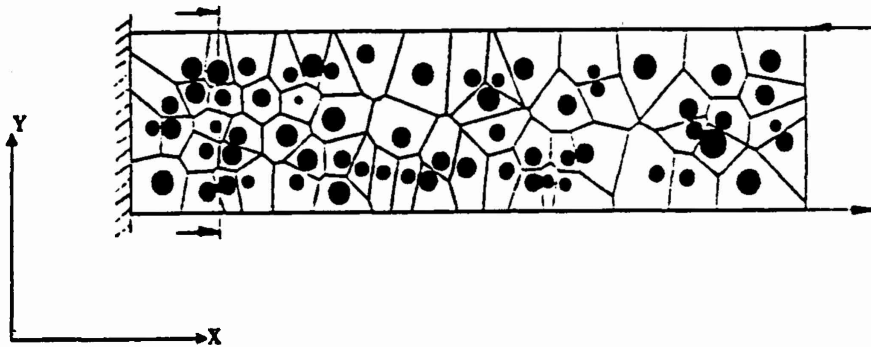


Fig. 34. Composite beam subjected to pure end moment (64 elements)

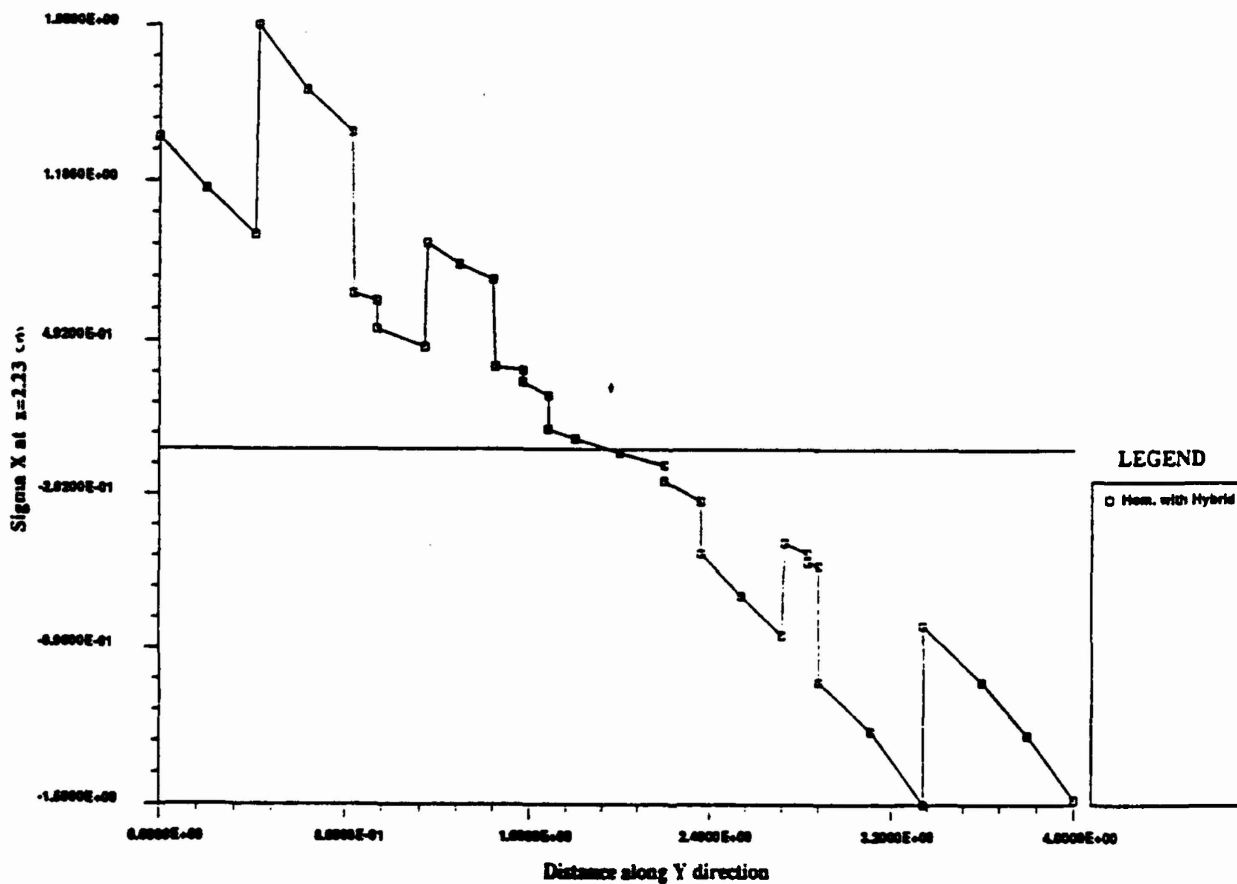


Fig. 35. Sigma X across a section

5 References

1. R. L. Mallett, Voronoi Cells as Finite Elements, ALCOA Report.
2. T. H. H. Pian, Derivation of Element Stiffness Matrices by Assumed Stress Distribution, AIAA Journal, Vol. 2, No. 7, July 1964, pp. 1333-1336.
3. Robert D. Cook, Improved Two-Dimensional Finite Element, Journal of The Structural Division, ASCE, No. ST9, September 1974, pp. 1851-1863.
4. M. L. Accorsi, A Method for Modelling Microstructural Material Discontinuities in A Finite Element Analysis, Int. J. Num. Meth. Engng., vol 26, 2187-2197 (1988).
5. Noboru Kikuchi, Finite Element Methods in Mechanics, Cambridge University Press, 1986